

S7+ Portal Frame

Contents

Application options	2
Basis of calculation	4
Basic operating instructions	5
Interactive 3-d graphical user interface	5
Wizard	6
Definition of the structural system	7
Basic parameters	7
Structural system	9
Cross sections	10
Haunches	12
Roof / Wall construction	14
Border conditions	14
Loads	16
Standard load cases	17
Additional load cases	17
Input of bracket loads on the frame column in the graphic	19
Verifications and design	21
Ultimate limit states	21
Serviceability limit states	21
Max / Min shear values	22
Design of the connection details	22
Output	24

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 S7+ application is suitable for the calculation of single-bay portal frames.

The available roof types are

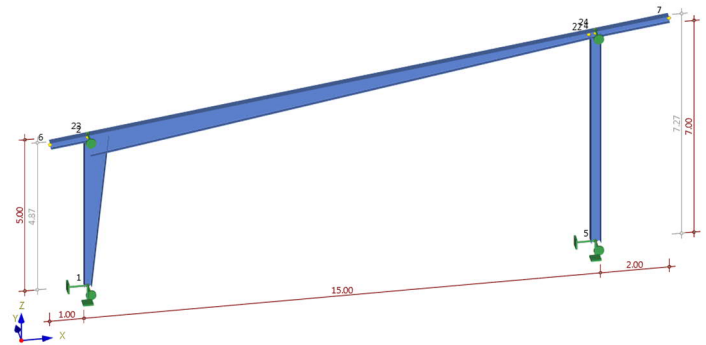
- double-pitch,
- single-pitch and
- flat roofs.

Roof projections and asymmetrical frames can also be defined.

The software generates the structural system from the specified outer dimensions, roof shape and cross sections.

By entering a few parameters, such as roof shape, outer dimensions, cross-sections and location (municipality), the structural system and the necessary wind and snow loads are generated by the program.

Dimensions can optionally be related to outer or system dimensions.



The column bases can be defined as pinned or restrained, whereby spring values can also be entered for horizontal displacement and restraint.

Connections in the frame corners and the ridge can be pinned or rigid or by defining torsion springs, if the rotational stiffness in the respective connection is known.

Haunches can be defined on the horizontal and vertical members as well as cantilevers in the area of the frame corners.

Available standards

The S7 + program carries out the structural safety verifications according to EN 1993-1-1 taking into account the regulations of the national annexes.

- DIN EN 1993-1-1/NA
- ÖNORM B 1993-1-1

The standards EN 1991-1-3 and EN 1991-1-4 apply for snow and wind loads, taking into account the regulations of the National Annexes

- DIN EN 1991-1-3 / NA; DIN EN 1991-1-3 / NA
- ÖNORM B 1991-1-3; ÖNORM B 1991-1-4

Verifications

The internal forces are determined with the help of the elastic frame method. The load combinations decisive for the design are calculated in a second-order analysis with consideration of the initial sway imperfection.

All necessary combinations of actions are automatically taken into account in accordance with the safety concept set forth in DIN EN 1990.

The verification of the cross-sectional resistance is based on the internal plastic limit forces. You can optionally select the theory of elasticity as a verification method.

In the examination of components perpendicular to the frame plane, you can take lateral supports, rotational and translational beddings, e. g. bracings, purlins or trapezoidal sheets, into account.

The software calculates the deformation of the structural system as well as the relative deformation of the individual components in the serviceability limit state in accordance with the selected design situation.

The support reactions are put out separately for each load case by indicating the characteristic loads and/or the design loads resulting from the second-order analysis.

Load transfer and interfaces

There are interfaces to the programs for the design of the connections:

SPS+ Butt Plate Joint

SRE+ Frame Corners Steel

BTII+ Detailed (extended) verifications of stability with the application Lateral Torsional Buckling Analysis
BTII+.

Load transfer to

ST3 Steel Column Base and

ST6 Pocketed Steel Column Base.

FD+ Isolated Foundation

FDB+ Block Foundation

Basis of calculation

The calculation is based on the displacement method. The horizontal and vertical node displacements and the torsion are the unknowns of the structural system.

Deformation caused by bending and axial force is taken into account. In the second-order analysis, the actual cross-sectional properties must be used, because displacement is included in the global stiffness of the system via so-called geometric stiffnesses. The geometric stiffness takes the balance of the deformed system into account.

Basic operating instructions

Help texts and information about each individual definition value are an integral part of the user interface.

Clicking in an input field displays a description of the corresponding value in the information area.

A general description of the user interface is available in the document:

[Basic Operating Instructions-PLUS](#)

Interactive 3-d graphical user interface

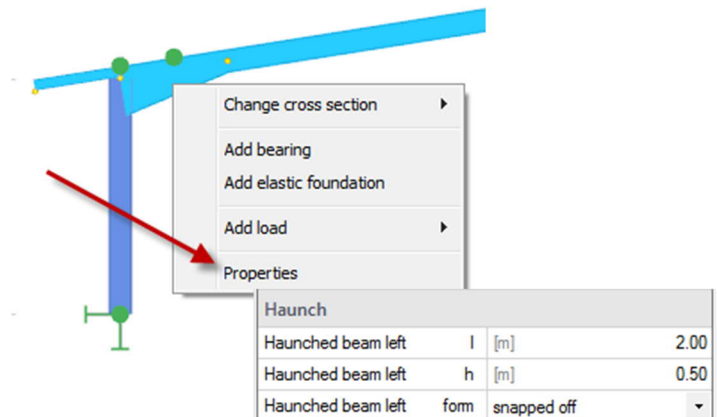
The central feature is the three-dimensional graphical user interface. All entered values are immediately shown on the graphic screen true to scale. When you keep the mouse cursor on a graphical object for a short while, a tooltip with information about the properties of the object is displayed. Clicking on a value enables the associated input field.

See also Basic Operating Instructions ▶ [Graphical Definition Options](#)

Important note concerning the context menu:

To display the context menu, right-click on an object (crossbeam, column, support, load, etc.). As the displayed menu is context-sensitive, it always provides the suitable options to edit or process the active object and provides access to the corresponding functions, fields and dialogs.

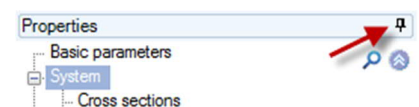
Operations like adding a support, defining loads and load cases or editing the properties of a haunch can be handled quickly and easily by right-clicking and selecting the corresponding option (see illustration on the right).



Effective workflow:

S7+ is conceived in such a way that you can define the entire structural system with the help of [assistants](#) and the interactive graphical user interface, i. e. via the context menu accessible per right click.

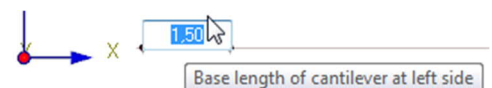
You can hide or display the definition area on the left of the screen via the Auto Hide button (see illustration on the right).



Editing dimensional values on the graphic screen

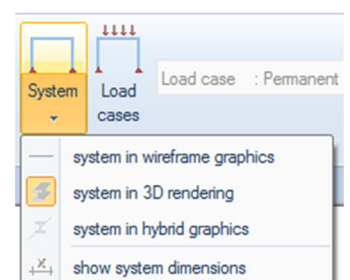
User-defined dimensions and load ordinates can be edited with a left click.

To confirm a changed value, press Enter or click left outside of the input field. The changed value is saved and the graphical representation is adjusted accordingly.



View - graphics - selection of the representation

This menu allows you to select the contents of the graphical representation, the representation size (zoom, pan, etc.), the font size, the load case to be represented. You can toggle between the representations of the system and the loads.

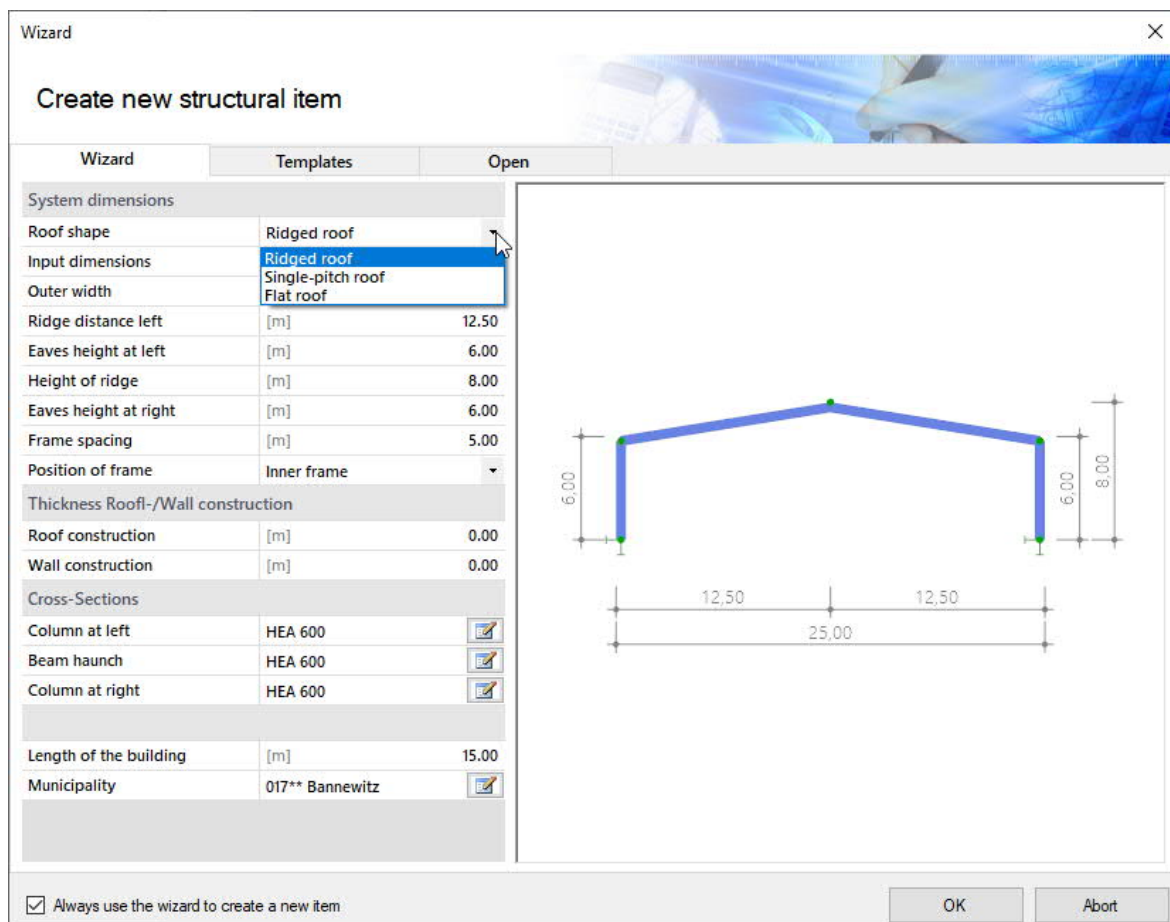


Wizard

The wizard is launched automatically when you start the application program. You can enter quickly the most important key figures of the frame system in the displayed window. These values can be edited subsequently in the definition area or on the [interactive graphic screen](#).

The assistant offers three frame systems for selection. You can define the cross sections and determine wind and snow load zones by specifying the location of the structure.

The application program generates the wind and snow loads on the frame system automatically. The load cases are matched to each change that you apply subsequently to the structural system.



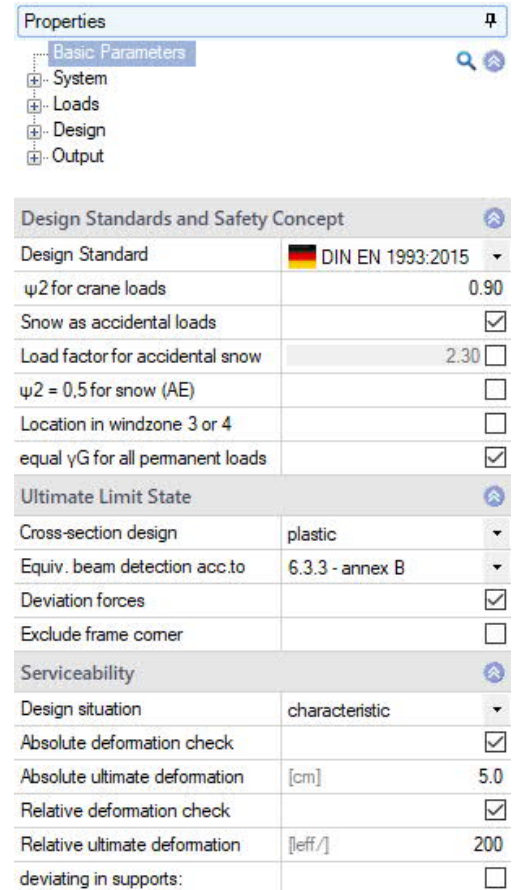
Note: You can disable the automatic start of the assistant via the corresponding option on the bottom of the window.

Definition of the structural system


Basic parameters

Standard and safety concept

Design Standard-	Definition of the design standard and its National Annex
ψ_2	Specifies the load combination factor ψ_2 for crane loads (= ratio of permanent part to total crane load)
Snow as accidental loads	If ticked in addition to all ordinary design situations snow loads are also considered as accidental loads.
Load factor for accidental snow	Defines the load factor for snow loads in the accidental design situation.
$\psi_2=0.5$	Specifies that the combination factor ψ_2 for snow in case of the earth quake design situation has to be raised to a value of $\psi_2 = 0.5$.
Location in windzone 3 or 4	Specifies that the construction is located in windzones 3 or 4. In this case snow needs not to be considered as accompanying action together with wind as leading action.
Consequence class	Only with ÖNORM: specifies the reliability level (in terms of the respective consequence class) of the partial safety factors.
Equal γ_G for all permanent loads	Defines whether equal values of the partial safety factor ($\gamma_{G,sup}$ or $\gamma_{G,inf}$) should be assumed for all permanent loads/loadcases. Otherwise the permanent loads/loadcases will be combined among each other with $\gamma_{G,sup}$ and $\gamma_{G,inf}$.



The screenshot shows the 'Properties' dialog box with the 'Basic Parameters' section expanded. The 'Design Standards and Safety Concept' section is highlighted, showing the following settings:

Design Standards and Safety Concept	
Design Standard	 DIN EN 1993:2015
ψ_2 for crane loads	0.90
Snow as accidental loads	<input checked="" type="checkbox"/>
Load factor for accidental snow	2.30 <input type="checkbox"/>
$\psi_2 = 0.5$ for snow (AE)	<input type="checkbox"/>
Location in windzone 3 or 4	<input type="checkbox"/>
equal γ_G for all permanent loads	<input checked="" type="checkbox"/>
Ultimate Limit State	
Cross-section design	plastic
Equiv. beam detection acc.to	6.3.3 - annex B
Deviation forces	<input checked="" type="checkbox"/>
Exclude frame corner	<input type="checkbox"/>
Serviceability	
Design situation	characteristic
Absolute deformation check	<input checked="" type="checkbox"/>
Absolute ultimate deformation	[cm] 5.0
Relative deformation check	<input checked="" type="checkbox"/>
Relative ultimate deformation	[eff./] 200
deviating in supports:	<input type="checkbox"/>

Structural safety

Cross-section design	Cross-sectional design can be elastic according to equation 6.1 or plastic according to equation 6.2.
Equiv.beam detection acc. to	The equivalent bar analysis is carried out according to 6.3.3 (Annex A o B) or 6.3.4
Deviation forces	If the option is ticked, the deflection forces that act between adjacent components as a result of the different direction of the member axes are calculated in the equivalent bar model within a set of members.
Exclude frame corner	When you tick this option, the frame corner is excluded in the verification of the cross section's load-bearing capacity. No design on the idealized member section is performed inside the shear field.

Serviceability

Design situation:	Verification of the serviceability in the characteristic, frequent and quasi-permanent design situations.
Absolute deflections:	the verification of the stability against deformation is based on the undeformed system.
Relative deformation:	the verification of the components' stability against deformation is based on the effective lengths, which are determined by the inflexion points of the bending line.
Deviating in supports	Tick this option if you want to enter different limit values for the columns - the input fields are then displayed.

Structural system

Input / selection of material, topology and system dimensions

System dimensions

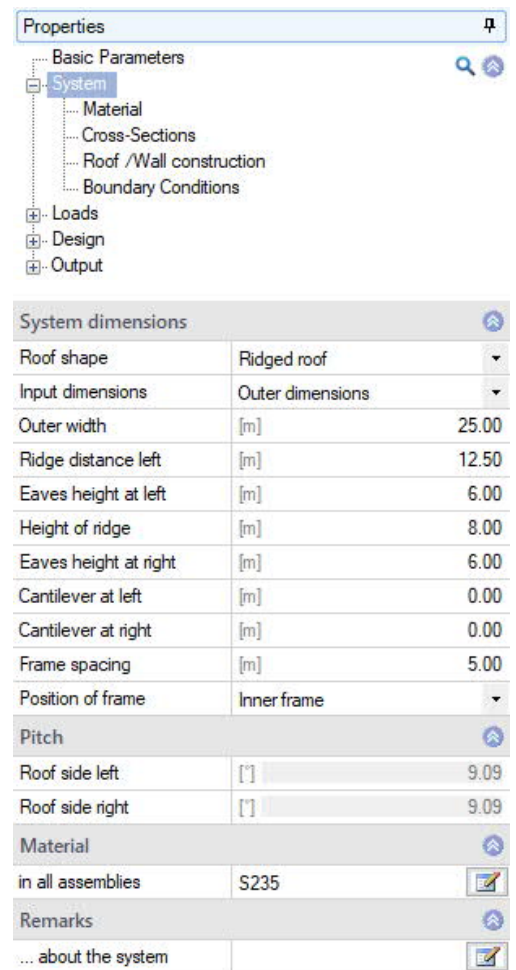
The input of the dimensions can either refer to outer or system dimensions. If dimensions are changed, this leads to the adaptation of dependent values by the program. Based on the selection, the entered outer dimensions or the system dimensions are fixed.

When entering the dimensions via the outer dimensions, the global reference point is on the outside of the left column including the thickness of the wall structure at the height of the column base. If the dimensions are related to system dimensions, the global reference point is in the system axis of the left column at the height of the column base. By changing between outer dimensions / system dimensions, the reference of the bearings and loads entered is automatically adjusted by the program.

Regardless of this, the value of dimension chains that relate to outer or system dimensions can be changed within the graphic at any time.

Material

The material for the beam and column can be selected separately if necessary.



The screenshot shows the 'Properties' panel in a software application. The 'System' section is expanded, showing a tree view with the following items: Basic Parameters, System (selected), Material, Cross-Sections, Roof /Wall construction, Boundary Conditions, Loads, Design, and Output. Below this, the 'System dimensions' section is expanded, displaying a table of parameters and their values. The 'Pitch' section is also expanded, showing roof side dimensions. The 'Material' section shows 'in all assemblies' set to 'S235'. The 'Remarks' section has a text entry '... about the system'.

System dimensions	
Roof shape	Ridged roof
Input dimensions	Outer dimensions
Outer width	[m] 25.00
Ridge distance left	[m] 12.50
Eaves height at left	[m] 6.00
Height of ridge	[m] 8.00
Eaves height at right	[m] 6.00
Cantilever at left	[m] 0.00
Cantilever at right	[m] 0.00
Frame spacing	[m] 5.00
Position of frame	Inner frame
Pitch	
Roof side left	[°] 9.09
Roof side right	[°] 9.09
Material	
in all assemblies	S235
Remarks	
... about the system	

Cross sections

The software currently handles all double-symmetrical steel shapes (double T) of the FRILO profile selection file as well as I-shapes with user-defined dimensions in their normal position.

Click on the edit-button to access the [cross-section selection](#).

To enter user-defined dimensions (height, width, etc.),

- select a profile from any shape group,
- click the Edit-button (the selected profile is used as the starting profile for the user-defined profile),
- adjust the dimensions and define a suitable name,
- save the new profile with OK.

Your own profiles can be saved in the Frilo profile database so that they are also available in other programs (button "Save to Database").

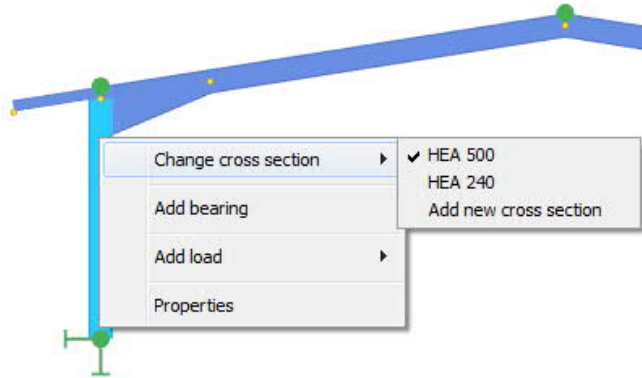
You can also display the "Static values" and define individual profiles as "Favorites".

Name	IY cm4	IZ cm4
180	2510	925
200	3690	1340
220	5410	1960
240	7760	2770
260	10500	3670
280	13700	4760
300	18300	6310
320	22900	6990
340	27700	7440
360	33100	7890
400	45100	8560
450	63700	9470
500	87000	10400
550	112000	10800
600	141000	11300
650	175000	11700

Alternative definition options in the interactive 3-d graphical interface

Right-click on the desired component to display the context menu.

Edit cross section: you can either select one of the currently used cross sections or add a new one.



Haunches

You can define haunches on the horizontal and vertical members and eaves overhang haunches to brace the frame corners.

The available haunch shapes are "Stiffener plate and flange" or "snapped off/kinked flange" (widened beam). Column haunches are assumed over the entire length of the column.

Alternative definition options via the 3-d graphical user interface are accessible via the context menu ▶ Properties.

Crossbeam haunch

The diagram shows a crossbeam haunch with dimensions l and h . The 3D model shows a beam with a haunch and a context menu with the following options:

- Change cross-section
- Add support
- Add elastic foundation
- Add load
- Design using BTH+
- Properties

Properties	
Pitch	
Roof side left	[°] 9.09
Cross-Sections	
Beam haunch	HEA 600
Material	
in all assemblies	S235
Haunch	
Hunched beam at left	l [m] 2.00
Hunched beam at left	h [m] 0.50
Hunched beam at left	Shape snapped off
Restraint of Beams	
Elastic restraint of upper flange	[kN/m ²] 0.00
Rotational foundation	[kNm/rad/m] 0.0

Column haunch

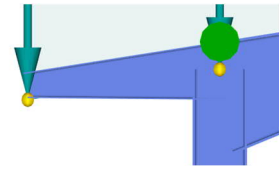
The diagram shows a column haunch with dimensions h and h_{Profil} . The 3D model shows a column with a haunch and a context menu with the following options:

- Change cross-section >
- Add support
- Add load >
- Design using BTH+
- Properties

Haunch	
Hunched column left	h [m] 0.50
Hunched column left	Shape snapped off
Shape of beam haunch: Profile (internal chord) or standard CS + Stiffener plate and flange	

Eaves overhang (Cantilever) haunches

Same as above - see graphic on the right side.



Roof / Wall construction

To improve the model accuracy, the thickness of the roof structure from the outer edge of the beam or the thickness of the wall structure from the outer edge of the column can be entered as the distance to the outer edge of the building envelope structure. This parameters affects the determination of the required wind and snow loads and must be taken into account in the global reference point.

Thickness Roof-/Wall construction		
Roof construction	[m]	0.00
Wall construction	[m]	0.00

Border conditions

The column bases (base points) can be defined differently - pinned or restrained, whereby spring values can also be entered for horizontal displacement and clamping.

The frame corners and the ridge points are pinned or rigid.

Their rotational stiffness can, however, also be taken into account by entering torsion springs if these are known after the connections have been calculated. If a frame corner is modeled rigidly, the column can be connected to the beam in an pinned manner or with a spring value.

- fixed in x + hinged about y
- fixed in x + restrained about y
- Properties

Supports		
Distance	[m]	0.30
Spring value	Cy	rigid <input checked="" type="checkbox"/>
Impact point	Cy	Member centroid
Spring value	Cpx	Upper edge
		Member centroid
		Lower edge

Point of impact Cy in CS [edge / member axis / bottom edge]

III.: Alternative definition of the border conditions in the 3-d GUI

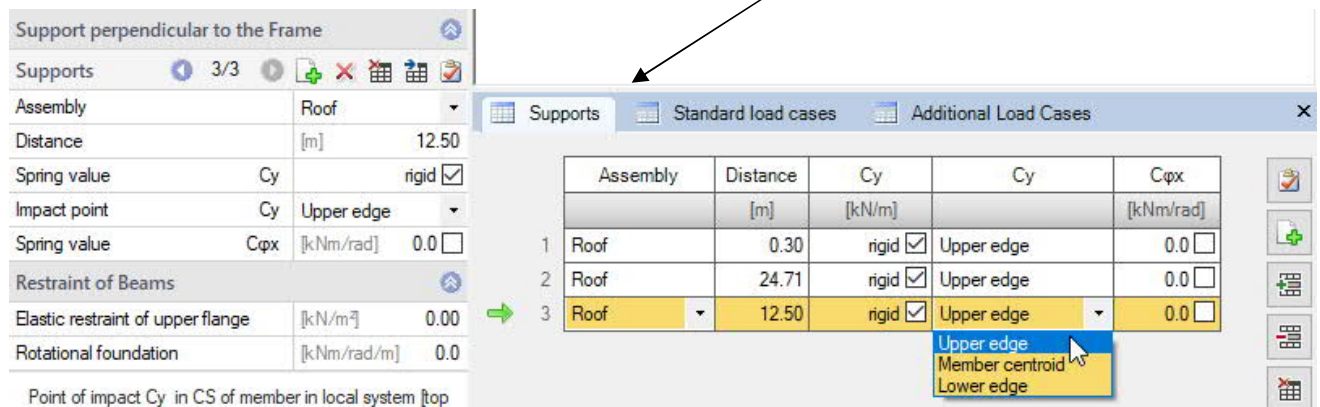
Supports out of plane

In the examination of components perpendicular to the frame plane, you can take lateral supports, rotational and translational beddings, e. g. bracings, purlins or trapezoidal sheets, into account.

It is of essential importance for the examination of the stability to define where the lateral supports apply to the cross section.

Supports at the top or bottom chord or in the shear centre of the cross section are available.

You can define the individual supports either in the Supports table on the Supports tab or in the definition area via the [List entry](#).



Support perpendicular to the Frame

Supports 3/3

Assembly: Roof

Distance: [m] 12.50

Spring value: Cy rigid

Impact point: Cy Upper edge

Spring value: C_{φx} [kNm/rad] 0.0

Restraint of Beams

Elastic restraint of upper flange: [kN/m²] 0.00

Rotational foundation: [kNm/rad/m] 0.0

Point of impact C_y in CS of member in local system [top]

	Assembly	Distance [m]	C _y [kN/m]	C _y	C _{φx} [kNm/rad]
1	Roof	0.30	rigid <input checked="" type="checkbox"/>	Upper edge	0.0 <input type="checkbox"/>
2	Roof	24.71	rigid <input checked="" type="checkbox"/>	Upper edge	0.0 <input type="checkbox"/>
3	Roof	12.50	rigid <input checked="" type="checkbox"/>	Upper edge	0.0 <input type="checkbox"/>

Dropdown menu options: Upper edge, Member centroid, Lower edge


Loads

Self-weight

The self-weight of the frame is automatically determined when this option is ticked.

Roof load

Consider roof load permanent loads caused by the roof structure.

Roof load g Enter the load value g . A load value compilation can be called up using the "arrow symbol"  - see description in the LOAD+ program.

Roof load based on Relation of the load from the roof structure to the roof or the base area.

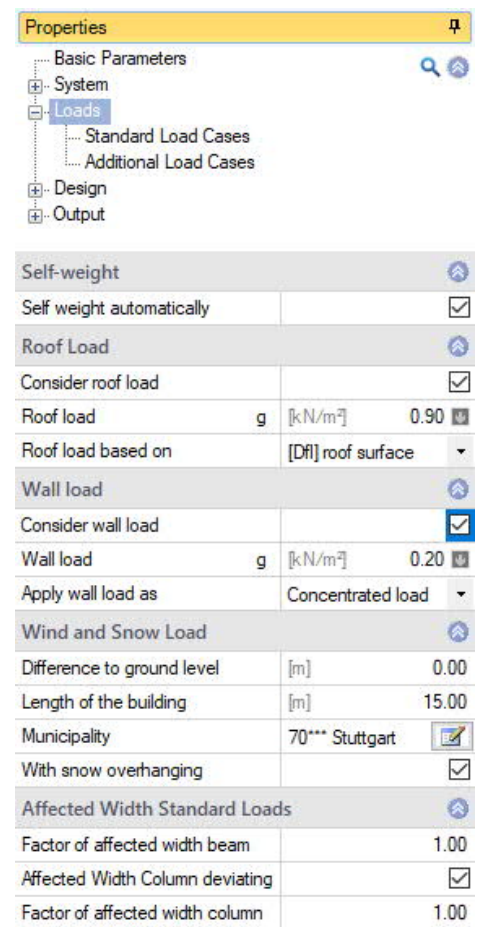
Wind and Snow Load

Difference to ground level Height of the frame base above the ground.

Length of the building Building length transverse to the wind attack from the left / right for the division of the wall and roof areas for the determination of the external pressure coefficients.

Municipality Selection of the wind and snow load zone. The list of the DIBt (German Center of Competence of Civil Engineering) is available for the assignment of the location to a wind and snow zone. See [Wind and snowloads](#).

With snow overhang Calculate snow with overhang at the cantilevers.



Affected Width Standard Loads

Factor of affected width The affected width factor allows you to consider the continuity effect of the roof structure in the calculation of the standard load cases.
 ... deviating: The affected width for the standard loads can be entered in the area of the columns that differs from the affected width of the beams.

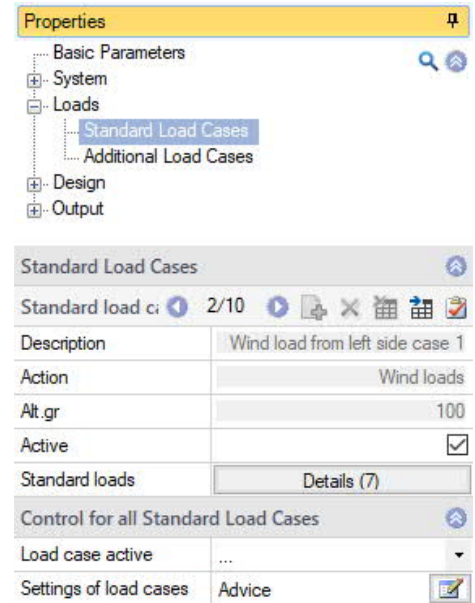
Standard load cases

The standard load cases include the "permanent" loads, and the wind and snow loads generated by the software.

They are generated by the program via the building dimensions, snow and wind zones. They cannot be edited and are created anew each time the system is changed.

You can disable these load cases via the "Active" option.

You can use the Details button to display the values for this load case in a table.



Control for all Standard-Load Cases

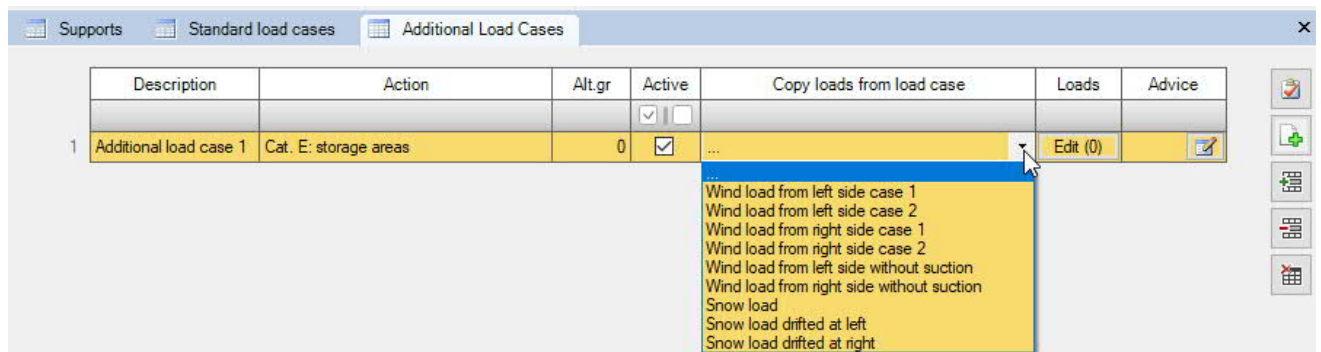
Load case active With this selection list you can optionally switch different load case categories on or off. Inactive load cases are not taken into account in the calculation.

Additional load cases

If additional loads are required, you can define this via the "Additional Load Cases" tab. You can copy loads from the standard load cases as a template and adjust them using the "Edit" button.

You can create new table lines for additional load cases using the symbol

See also: [Table input](#)



Load input für additional load cases

You can define the loads either via the "Edit" button / dialog or directly in the interactive graphic.

Entering and changing loads via the edit dialog

You can create new table lines for additional load cases using the symbol

Loads										
	Assembly	Load type	Description	Dirrection	Load reference to	Value begin	Value end	Distance	Length	Impact point
1	Wall_at_left	Line load	Additional load 1 [Wi	--	--	1.77	1.77	0.00	5.75	Upper edge
2	Wall_at_right	Line load	Additional load 2 [Wi	--	--	-0.80	-0.80	0.00	5.75	Upper edge
3	Roof	Line load	Additional load 3 [Wi	[Df]	roof surface	-2.59	-2.59	-1.98	1.50	Upper edge
4	Wall_at_left	Line load	Additional load 4 [Wi	[Df]	roof surface	-1.19	-1.19	-0.48	12.98	Upper edge
5	Wall_at_right	Line load	Additional load 5 [Wi	[Df]	roof surface	-1.91	-1.91	12.50	1.50	Upper edge

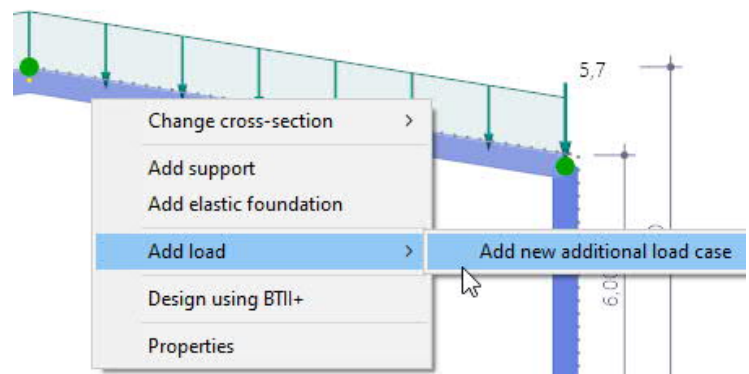
Assembly Here you choose whether the load acts on the roof or the walls on the left / right.

Load type Line- or Concentrated load, Moment.

Description	If necessary, you can enter a name here.
Direction	Roof: Line- or Concentrated load orthogonally on the base or roof surface, Concentrated load also in the axial direction of the beam
	Wall: Line loads orthogonally on the wall surface Concentrated loads across the column = acts orthogonally on a component in the local system or in direction of the column = Columns positive downwards, beam positive to the right
	You also enter the load value (value begin), for line load the start and end value as well as the length. If necessary, you can call up the load value compilation using the small load symbol.
Distance	Distance from the reference point of the assembly Roof: Measured from the origin of the column on the outside left including the specified thickness from the wall structure with a distance in the horizontal direction. Wall: Measured from the origin in the base point with a distance in the vertical direction.
Impact point	Here you select the point of application of the load in the cross-section of the component in the local system: upper edge / component axis / lower edge.

Entering and changing loads in the graphic

The illustration on the right shows an example of the input options in the graphic if you click on the beam.



Create an additional load case

Right-click on the relevant component and select "Add new additional load case".

In the following dialog loads from existing standard and additional load cases can be copied / adopted here and then edited.

Note: you may have to make the load cases visible in the graphic (under "Display" in the upper menu ribbon).

Alternatively: Right-click on a free area in the graphic and select "Additional load case > Create".

Entering a load

Wählen Sie einen Lastfall aus (anklicken). Der gewählte Lastfall wird mit einem Häkchen markiert. Für diesen Lastfall können Sie nun Einzel-, Streckenlasten und Momente definieren/ändern – klicken Sie dann auf den entsprechenden Eintrag (Einzellast, Moment, Streckenlast) um die entsprechenden Werte zu editieren.

Select a load case (click). The selected load case is marked with a tick. For this load case you can now define / change individual loads, line loads and moments - then click on the corresponding entry (point load, moment, line load) to edit the corresponding values.

Calling up the load input dialog

Um eine vorhandene Last zu ändern, Doppelklicken Sie in die Lastfläche (in der Grafik) oder klicken Sie mit der rechten Maustaste in die Lastfläche und wählen Sie „Eigenschaften“.

To edit an existing load, double-click in the load area (in the graphic) or right-click in the load area and select "Properties".

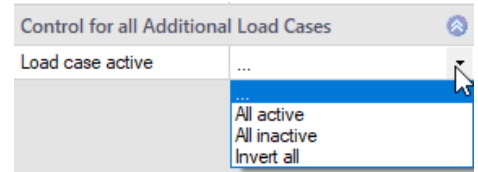
See also the example of load input on the following page.

Control for all additional load cases

With this function you can switch all additional load cases on or off together.

When "inverting" the settings are reversed:

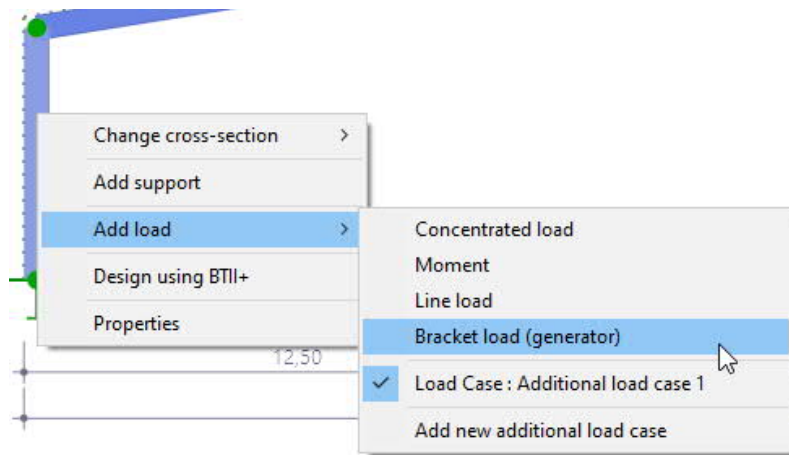
All previously inactive additional load cases are then active; Additional load cases that were previously active are then inactive and are not taken into account in the calculation.



Input of bracket loads on the frame column in the graphic

The console load generator can be used for more convenient input of loads resulting from a bracket.

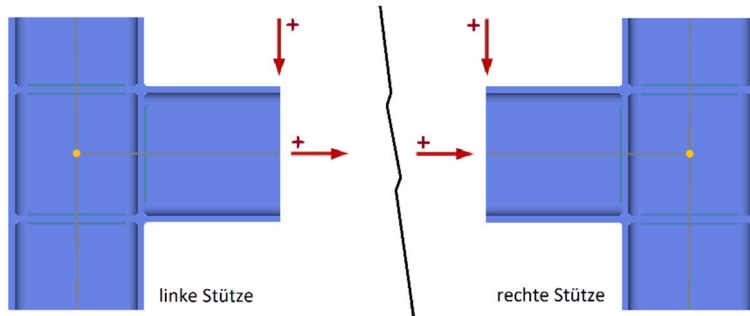
To do this, click a column with the right mouse button and select "Bracket load".



In the opened dialog, you can either define the load on the selected column or, at the same time, with identical dimensions on the opposite column.

Properties		?	×
Console inside			
Arrangement	on selected support		▼
clear length console	[m]	0.60	
Distance	[m]	3.00	
Origin		base point left	
Reference direction		vertical direction	
Loads			
Description		Console load left	
Force vertical	[kN]	80.5	
Force horizontal	[kN]	-10.1	
horizontal console load (positive to the right)			
		OK	

Sign of the bracket load input:



left column

right column

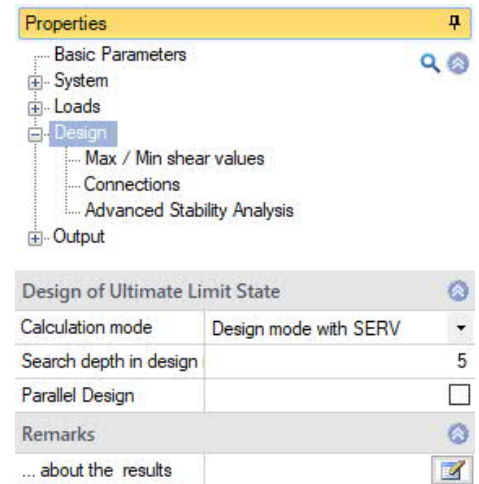
Verifications and design

Calculation mode

To shorten the computing time, two additional calculation modes can optionally be selected for the complete calculation for the design phase:

1. Design mode with evaluation of the calculation only in the limit state of the load-bearing capacity
2. Design mode with SERV, in which the serviceability is also evaluated.

The decisive load combinations are determined according to the first order theory and the internal forces are then only calculated for the decisive combinations according to the second order theory.



Ultimate limit states

The load combinations decisive for the design are calculated in a second-order analysis with consideration of the initial sway imperfection.

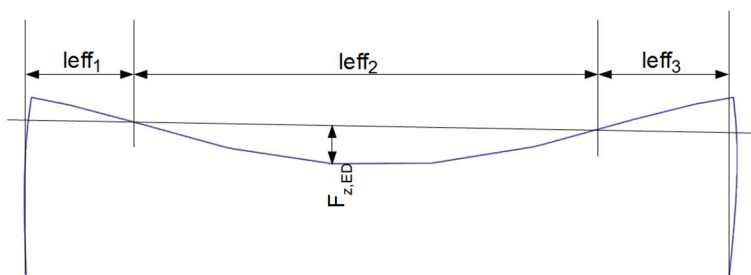
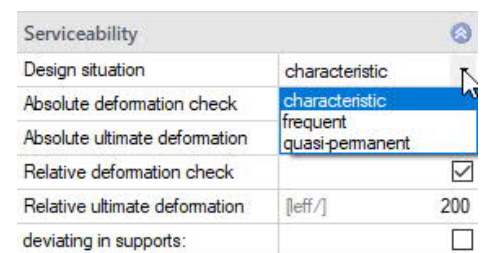
All necessary combinations of actions are automatically taken into account in accordance with the safety concept set forth in DIN EN 1990.

- Verification of the plastic cross-sectional resistance as per DIN EN 1993-1-1, para. 6.2.
If you have activated the "Elastic design" option when defining the [basic parameters](#), the elastic verification is performed in accordance with equation 6.1.
- Stability verifications of components as per DIN EN 1993-1-1, para. 6.3.

Serviceability limit states

A second-order analysis is performed for the design situation selected in the [basic parameters](#) section.

- Verification of absolute deflections
The serviceability verification is based on the difference between the deformed and the undeformed system.
- Verification of relative deflections
The serviceability verification is based on the difference in deformation referenced to the effective lengths. The effective lengths are determined by the inflexion points of the bending line.



Max / Min shear values

Here you can define additional positions for determining the Max / Min internal forces.

See also [output - results](#).

Design of the connection details

For the design of the connections, interfaces to the Frilo application programs

SPS+ Butt Plate Joint

SRE+ Frame Corners Steel

BTII+ Detailed (extended) verifications of stability with the application
Lateral Torsional Buckling Analysis BTII+.

Load transfer to

ST3 Steel Column Base and

ST6 Pocketed Steel Column Base.


FD+ Isolated Foundation

FDB+ Block Foundation

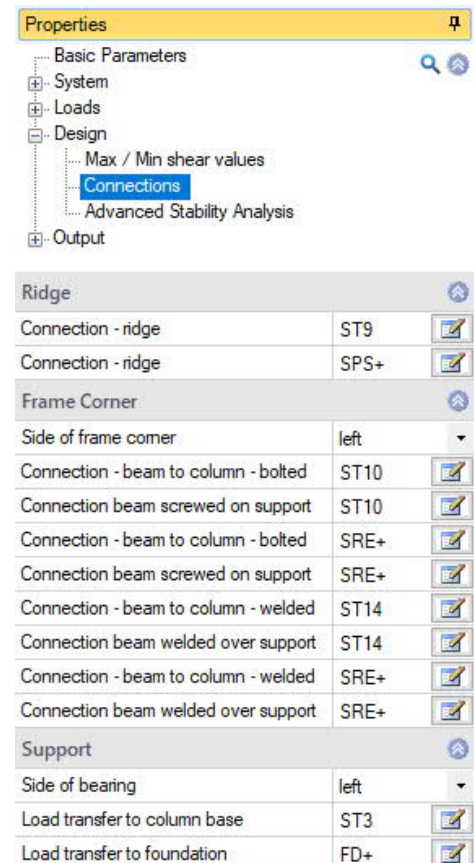
are available

These interfaces provide for the direct transfer of the geometry and the internal forces of the decisive design load cases to the corresponding application program. This program must be installed and licensed on the respective computer.

Note: the interfaces to the "old" programs ST9 / ST10 / ST14 will continue to work if the "new" programs are not licensed.

Click on the corresponding  button to launch the desired application or use the interactive graphic functions at the desired node.

Note: Changes in material and cross-section as well as the possible rigidity in the coupled programs SRE + / SPS + are returned to S7 +.



Ridge point connection – Butt Plate Joint SPS+ application

After having selected the node forces, SPS+ is launched.

Frame corner connection - Frame Corners Steel SRE+ application

Depending on the selected option, you can verify a screwed knee corner either with or without tension butt strap or welded connections can be verified with the SRE+ application (with estimation of an assembly butt plate thickness following a haunch).

Load transfer of the support reactions to the application programs:

ST3 – Steel Column Base and

ST6 – Pocketed Steel Column Base

The internal design forces according to the second order theory can be transferred or, alternatively, the characteristic support reactions per action.

After the transfer of the characteristic actions, the decisive design load cases are determined in the ST3 or ST6 application.

A transfer to ST6 is only possible with a restrained base.

Load transfer to foundation FD + and block foundation FDB +

The characteristic support loads for each load case according to the first order theory or, optionally, the internal design forces according to the second order theory can be transferred.

After the transfer of the characteristic support loads, the decisive design load cases are determined in the programs FD + or FDB +.

The transfer to the FDB + program is only possible when the base point is clamped.

Advanced Stability Analysis – interface BTII+

If the program BTII+ (Lateral Torsional Buckling Analysis) is licensed, the system can be transferred to BTII+. With BTII+, more complex systems can be calculated.

Output

Results

By checking the desired options, you can determine the scope of data to be put out. Activate the desired output options.

The results are put out in the form of tables. If several design situations have been handled, a separate table is put out for each design situation and each verification.

In the graphical output, always the decisive verification (with the highest utilization) is displayed if several sets of internal forces apply.

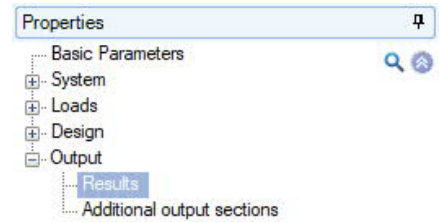
Additional output sections

Additional output sections for the decisive combinations are possible for the internal forces.

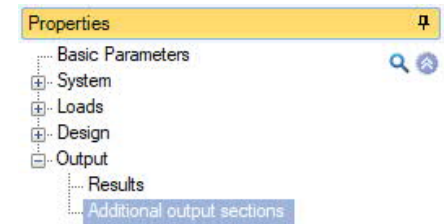
Output as a PDF file

The "Document" tab displays the document in PDF.

See also [Output and Printing](#)



Global	
Short Output	<input checked="" type="checkbox"/>
Results of Structural Elements	
Check of stability	<input checked="" type="checkbox"/>
Check serviceability	<input checked="" type="checkbox"/>
Reactions / Node Forces / Deformation	
Node forces II.o.	<input type="checkbox"/>
Reactions 1st.O. simple per Lc	<input checked="" type="checkbox"/>
Reactions 2nd.O.	<input type="checkbox"/>
Deformations	<input type="checkbox"/>



Additional points for the output of the internal fo		
Location	1/1	
Assembly	Roof	
Distance	[m]	0.00

Result graphs

On the "Results" tab, you can display the options and buttons for the representation of the result graphs.

