Single-span Steel Column STS+

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

Design standards

The *STS+* application performs structural safety analyses in accordance with the model column method for columns of steel profile sections as per EC 3 (EN 1993-1-1) under planned (ec-)centric loading. The regulations of the National Annexes are taken into consideration.

- DIN EN 1993-1-1:2010/ 2015
- ÖNORM EN 1993-1-1:2007/2017
- BS EN 1993-1-1:2008
- PN EN 1993-1-1:2010

Structural systems

The following structural systems are supported:

- Cantilever column
- Hinged column
- Column pinned on top and restrained on bottom
- Column restrained on top and on bottom
- General column (the supporting conditions can vary in the directions of the main axes)

Loads

You can apply vertical and horizontal loading on the column system and define moments. You cannot define loading that produces planned torsion, however. Moreover, you can define appended hinged columns in the directions of the both main axes and optionally apply the self-weight of the column.

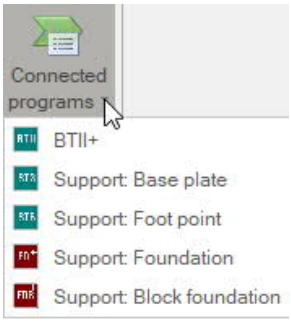
Calculation

STS+ generates automatically the appropriate load cases and load case combinations in accordance with the defined actions and performs the necessary analyses, whereby the decisive load case combination is determined for each limit state.

Interfaces to other applications

You can transfer the characteristic support reactions or the design values of the support reactions to the following software applications:

- FD+ Isolation Foundation
- FDB+ Pad Foundation
- ST3 Steel Column Base
- ST6 Pocketed Steel Column Base



If the real load conditions do not comply with the defined standard or the loading situation leads to planned torsion, you cannot use STS+ for the calculation. The *BTII+* application is available for this purpose.

If you have a valid licence for the *BTII+* (2nd Order Buckling Torsion Analyses) you can transfer the structural system from *STS+* to *BTII+* via the data export function. The *BTII+* application is also suitable for second order buckling torsion analyses of more complex systems.

See also Load transfer.

Basis of calculation

The basis of calculation of the *STS+* application is the Eurocode 3 standard series. The National Annexes for Austria and Great Britain are implemented in the current version of the application.

Design values of the internal forces

The calculation of the internal forces for the decisive load combination is performed in a first-order analysis.

All necessary combinations of actions are automatically taken into account in accordance with the safety concept set forth in the Eurocode 0.

The decisive internal forces combinations in the ultimate limit state are calculated for the verification of the cross-sectional resistance and the stability verification of the component.

The user must specify the design situation on which the serviceability analyses should be based.

The internal forces combinations for the design values of the support reactions are determined in addition.

Verification process

Analyses in the ultimate limit states

The load-bearing capacity verifications are based on the internal forces determined in a first-order analysis.

The stability verification of the component is based on the model column method. This analysis is preceded by a numerical calculation of the respective buckling load factors.

Analyses in the serviceability limit states

The serviceability verification refers exclusively to the calculation of the displacement, separately for the different main axis and the resultants.

Deformations are also calculated with internal forces determined in a first order analysis. You should keep in mind that deformations calculated in second-order analyses can be considerably greater in some cases. If the deformations are of particular importance, you should perform an advanced second order analysis. If you have a valid licence for *BTII+* you can use this application for this task

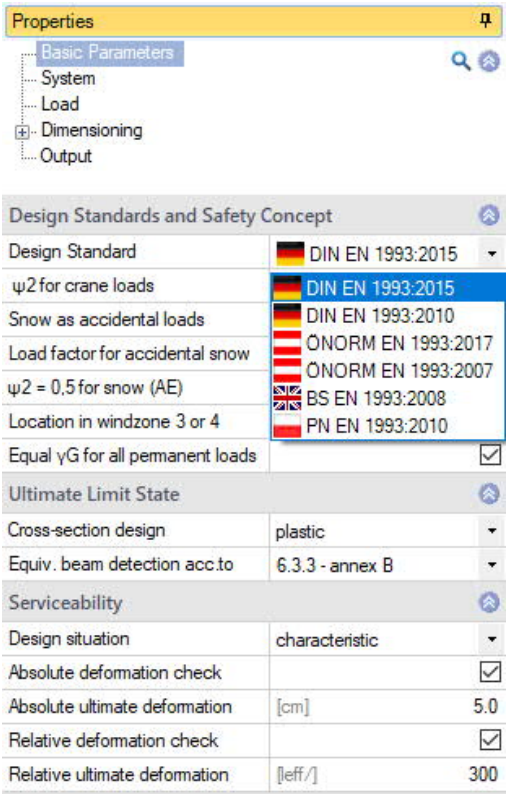
Load transfer

See Application options ▸ Interfaces

The supporting forces of the column system can be transferred to the applications Isolated Foundation (*FD+*), Pad Foundation (*FDB+*), Steel Column Base (*ST3*) and Pocketed Steel Column Base (*ST6*). You should note in this connection that the reaction forces are calculated in first order analyses.

Basic parameters

Standard and safety concept



Design standard — selection of the relevant National Annex for the load-bearing capacity verification as per EC3.

ψ2 for crane loads — Defines the combination coefficient ψ2 for crane loads (= ratio of permanent part to total crane load).

Snow as accidental loads — When you check this option, snow loads are also included as accidental action in addition to the typical design situations. The user can either specify a load factor for the accidental snow loads or have it determined automatically by the software.

ψ2 = 0,5 for snow — Check this option to increase the value of the combination coefficient ψ2 to 0.5 in the accidental design situation under snow load. (See introductory decree of the federal states, e.g. Baden-Württemberg)

Location in windzone 3/4 — Check this option if the building is situated in wind zone 3 or 4. In this case, you need not consider snow as an accompanying action with wind being the leading action.

Equal γG for permanent loads — Check this option if all permanent loads or load cases shall be considered with the same partial safety factor (γG,sup or γG,inf). Otherwise, all permanent loads or load cases are combined with each other with 'γG,sup' and 'γG,inf'.

Consequence class — allows you to define the consequence class the safety concept should be the based on: CC1, CC2 or CC3 (ÖNORM).

Structural safety

Cross section design — The cross section design is optionally performed in accordance with the
- elastic or the
- plastic method as per Para. 6.2

Model column verification — The verification in accordance with the model column method is based on 6.3.3 (annex A or B) or on 6.3.4

Serviceability

Design situation — defines the design situation for the verifications in the limit state of serviceability.

Absolute deformation check — performs the serviceability verification with consideration of the difference in deformation to the undeformed system.

Absolute ultimate deformation — the permitted maximum absolute imperfection of the structural system.

Relative deformation check — performs the serviceability verification with regard to the effective lengths, which are determined by the turning points (moment passage) of the bending line.

Relative ultimate deformation — the permitted maximum relative imperfection of the structural system.

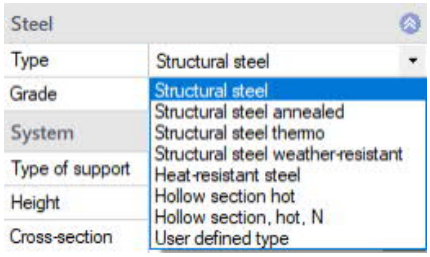
Structural system

Material

Steel type — this steel types (ill. right) are currently available for selection.

Steel grade — the available options for the steel grade depend on the selected steel type.

Parameters — if you have selected "user-defined type", you can display a dialog for the definition of the steel parameters by activating the  button. Otherwise, the parameters of the selected steel are displayed in this section.



Structural system

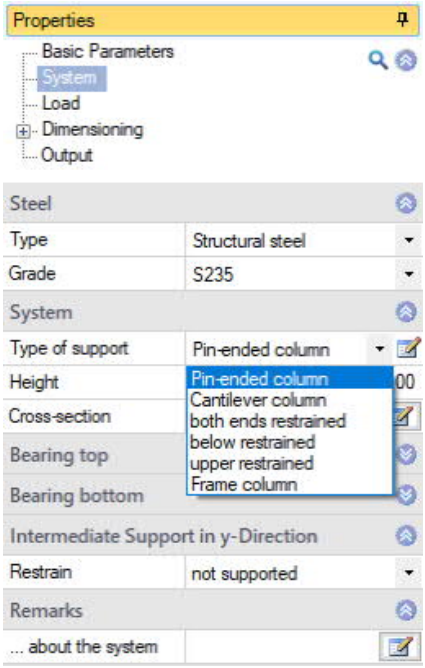
Column type: — selection of the column system. Activating the  button displays a selection dialog with graphical items.

Height — height of the column in the x-direction.

Cross section — activating the  button displays a dialog for the selection of the steel shape.

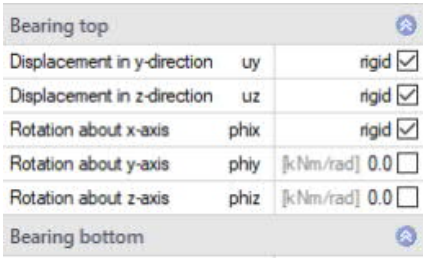
The manipulation of this dialog is described for all software applications in the document "Select - edit cross section".

Only steel shapes that are approved for the model column method are displayed.

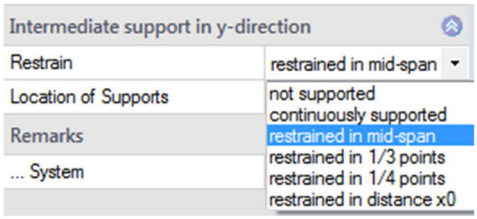


Bearing top/bottom

Displacement... — discrete supporting conditions for translation or rotation (in direction of/around the y- or z- axis):
Fixed: to enter a value remove the check mark:
0 = unsupported
> 0 = elastically supported



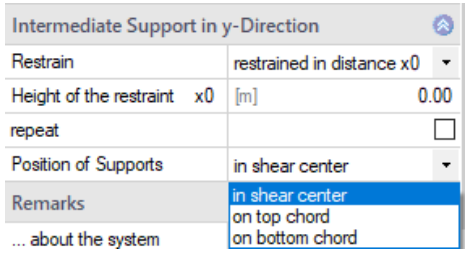
Intermediate support in the y-direction



You can define lateral fasteners in this section. This allows you to simulate applying bracing (discrete supports) or plate-type stiffening structures (continuous supports).

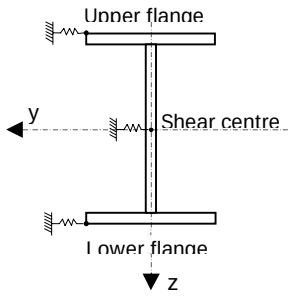
Note: *The supports are generated with a very high default spring value that produces a quasi rigid support. If you like to define more refined springs you should use the BTII+ application. (See interface to BTII+).*

Location of the support



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

The selection list allows you to specify the point of application of the lateral support.
See also the drawings:

Shear centre	
Upper flange (on top chord)	
Lower flange (on bottom chord)	

Remarks

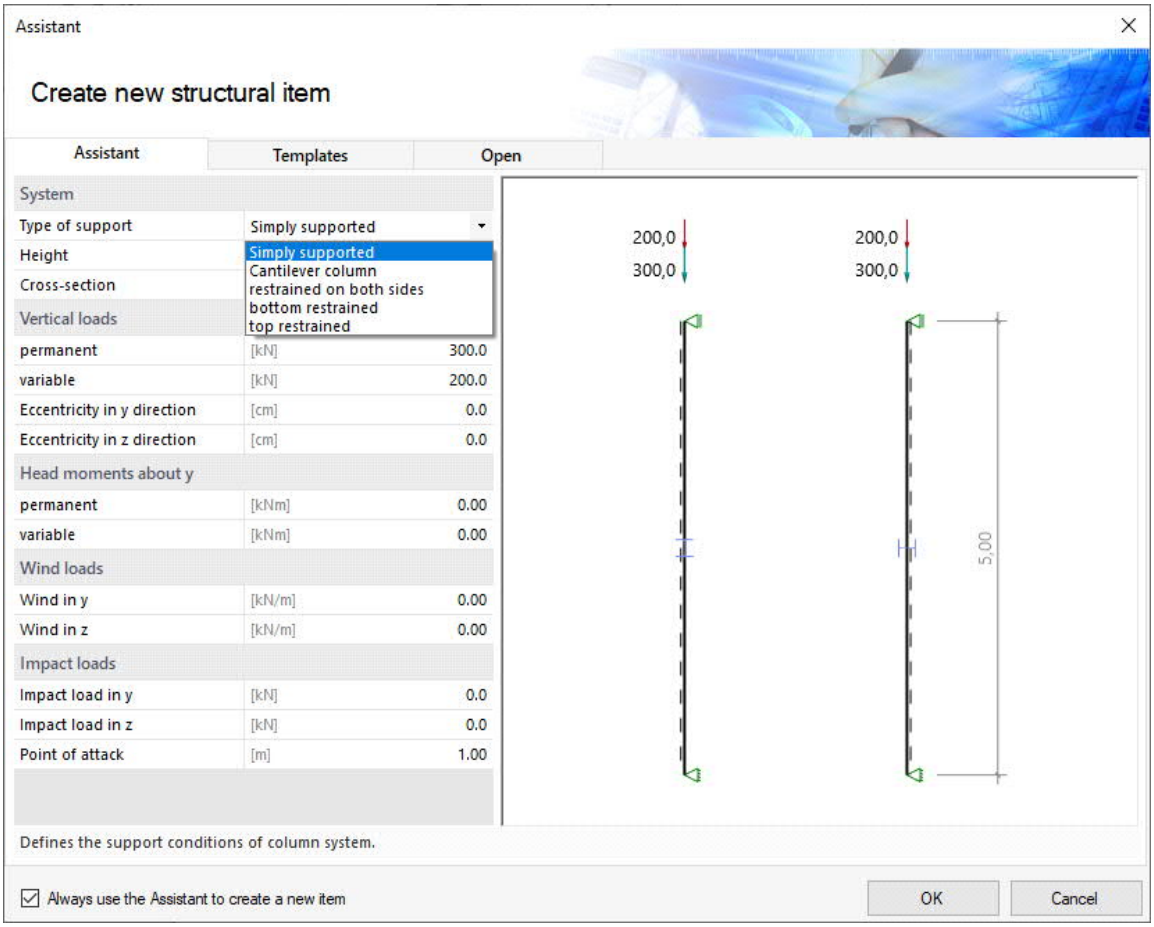
... about the system

activating the  button displays a dialog where you can enter an explanatory text. Optionally, you can display or hide this text in the output (the corresponding options are enabled when you enter a text).

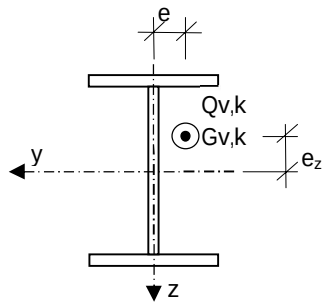
Loading

Standard loads / assistant

You can define the standard loads immediately after starting the program in the assistant. The assistant ensures that you can generate a calculable basic system by defining a few parameters. You can customize this basic system subsequently.



Vertical load

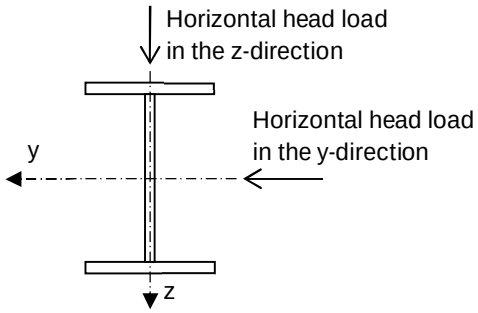
Value	Description	System sketch
Permanent	Permanent portion of the characteristic vertical load.	
Variable	Variable portion of the characteristic vertical load.	
Eccentricity	Eccentricity e_y /e_z of the load application point in the y-/z-direction (requires a sign)	

Action group

The vertical loads are always classified as "imposed loads of class A". The action can be edited later in the load table.

Head loads, horizontal

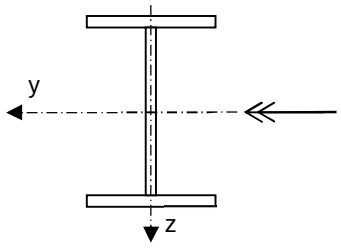
For cantilever columns

Value	Description	System sketch
Permanent in y-/z-direction	Permanent portion of the characteristic head load in the y-/z-direction.	
Variable in y-/z-direction	Variable portion of the characteristic head load in the y-/z-direction.	

Action group

The horizontal loads are always classified as " imposed loads of class A".

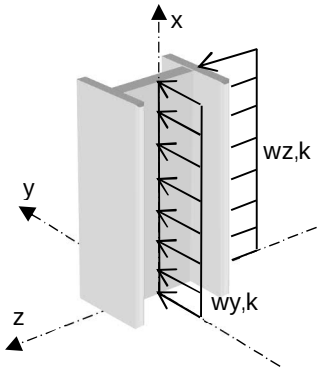
Head moments around y

Value	Description	System sketch
Permanent	Permanent portion of the characteristic head moment around the y-axis.	
Variable	Variable portion of the characteristic head moment around the y-axis.	

Action group

The node moments applying at the column head are always classified as " imposed loads of class A".

Wind loads

Value	Description	System sketch
Wind in y-direction	Characteristic value of the wind load in the y-direction.	
Wind in z-direction	Characteristic value of the wind load in the z-direction (*wz,k*).	

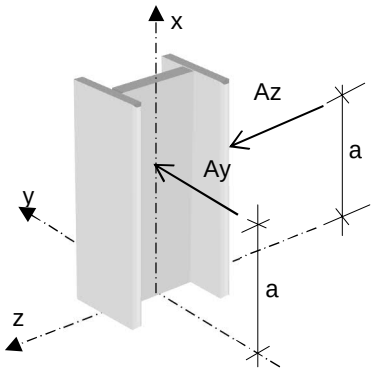
Action group

The wind loads are logically classified as "wind loads".

Alternative group

The wind loads are assigned to the first free alternative group (normally AltGrp=1), which means that they apply alternatively.

Impact loads

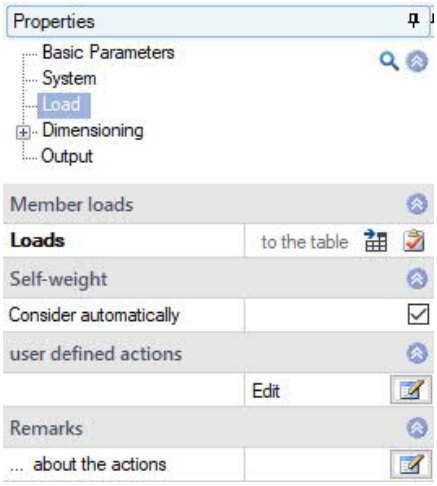
Value	Description	System sketch
Impact load	Nominal value of an accidental concentrated load in the y-/z-direction (Ay/Az)	
Application point	Application point *a* of an accidental concentrated load, measured from the base point.	

Action group

The impact loads are classified as "accidental actions".

Self-weight

...consider automatically — if you activate this option, the self-weight of the column is taken into account automatically.



User defined actions

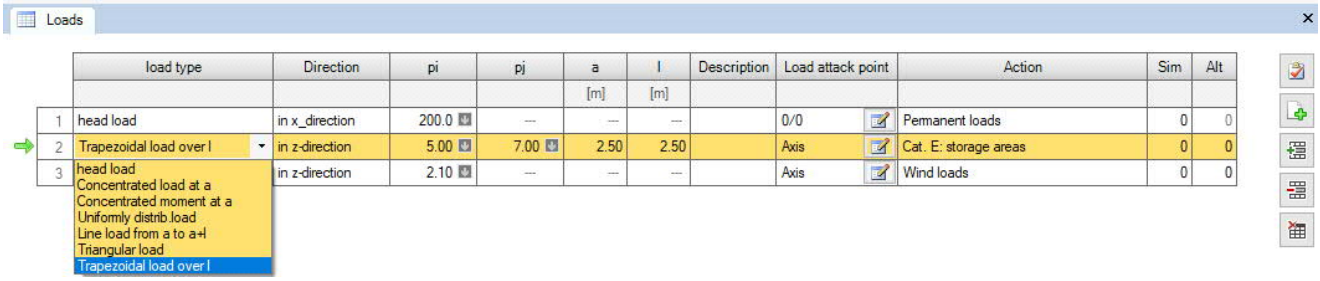
Input of actions in a dialog - see separate document.

Member loads

Loads

To the table: shows the load case table, which can also be displayed by activating the Loads button.

To define another load, insert a new row first by activating the button.



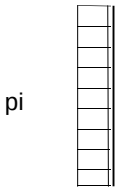
Load value compilation

By clicking on the arrow icon you can access a load value compilation.

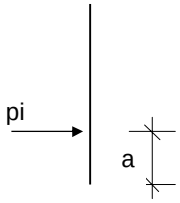
Tip: A description is displayed in the status line each time you click into a particular data-entry field.

Load type selection of a load type as described below. pi, pj are characteristic load values.

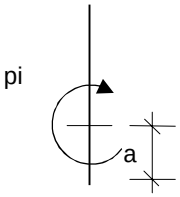
Head load	a load applying at the head/top of the column
Uniformly distributed load	a linear load that applies constantly over the total height of the column.



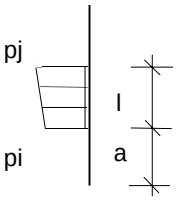
Concentrated load at a	a concentrated load applying at the distance *a* from the base point.



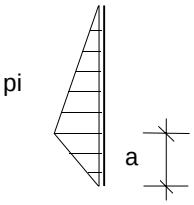
Concentrated moment at a	a moment applying at a distance *a*, measured from the base point



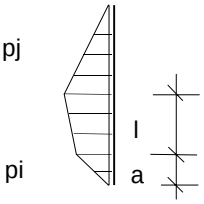
Line load from a to *a+l*	a linear variable line load applying over the column height, starting at a distance *a* measured from the base point and extending over a length *l*. Enter the load values for the front end and the rear end of the load extension.



Triangular load	variable triangular load applying over the entire column height.

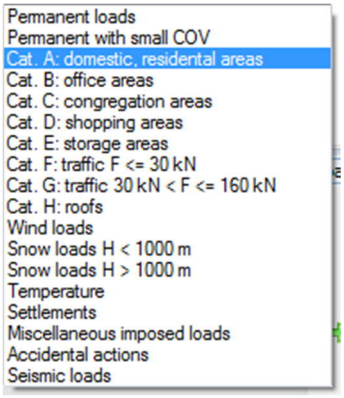


Trapezoidal load over l	variable trapezoidal load applying over the entire column height.

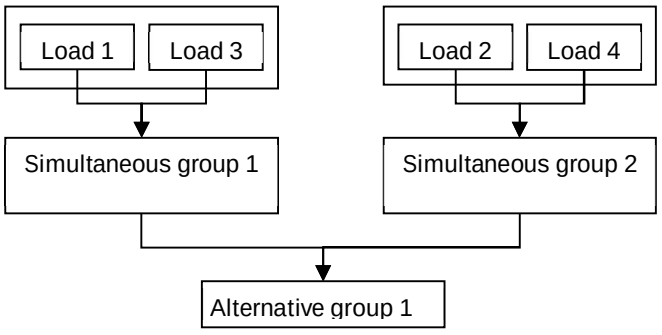


Direction selection of the direction of action. The loads or moments act in the direction of or around the global y-/z-axis. Concentrated loads also act in the direction of the x-axis.

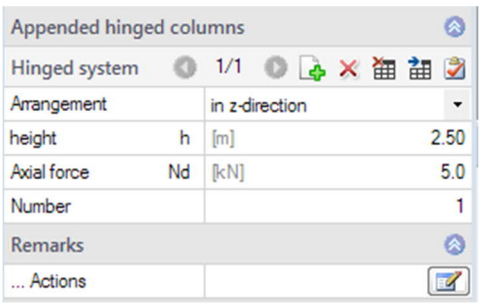
Description	you can optionally enter a short note that appears in the output.
Load attack point	selection of the load position on the cross section (top edge/bottom edge, component axis) or input of the y / z coordinates for the eccentricity of the load. You can display the corresponding dialog in the load table by activating the button.
Action	category or kind of action of the load See also user defined actions.
Simultaneous group	(Sim) assignment of the load to a group of loads acting jointly. The group is defined by a group number entered by the user. Loads that are assigned to the same simultaneous group always apply simultaneously. Loads in a simultaneous group must also be member of an action group.
Alternative group	(Alt) assignment of the load to a group of loads excluding each other. The group is defined by a group number entered by the user.
Remarks	allows you to enter personal remarks on the loads. You can optionally hide or display these comments in the output. The corresponding options are enabled when you enter a text.



Ill.:
Principle representation of the simultaneous and alternative groups.
Load 1 and 3 act together and are therefore assigned to the simultaneous group 1. The same applies to load 2 and 4 (simultaneous group 2).
The simultaneous groups 1 and 2 are assigned to the alternative group 1. Therefore, the loads of these two groups cannot apply simultaneously.



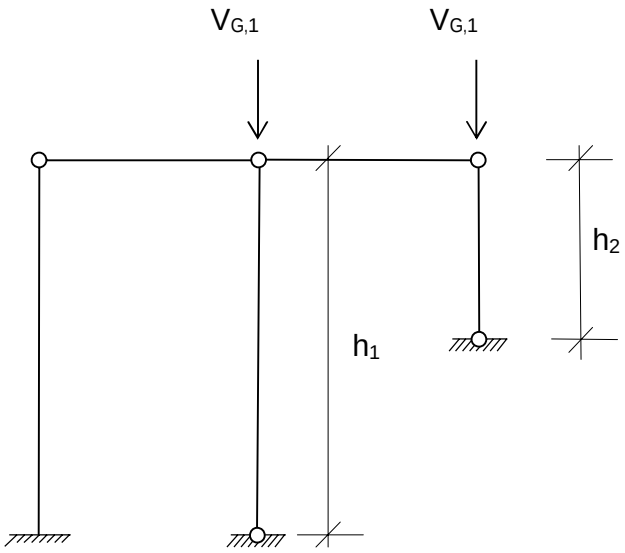
Appended hinged column



To handle cantilever columns, horizontal equivalent loads are generated for the appended hinged column.

(The entry fields only appear when the cantilever support is selected).

Arrangement	direction of action of the appended hinged column is the y-/z-direction
Height *h*	height of the appended hinged column
Axial force *Nd*	design value of the vertical load acting on the appended hinged columns.
Number	number of hinged columns appended in series



Remarks

... on the actions

The option displays a dialog for the input of text. You can optionally include this text in the output.

Design and analysis

Verifications in the ultimate limit state

Limit states

The analyses in the ultimate limit state include the following individual verifications:

- Analysis of the cross-sectional resistance with consideration of local buckling failure (verification of the c/t-limiting values and assignment to cross section classes).
- Verification of the plastic cross-sectional resistance as per EN 1996-1-1, para. 6.2.
  If you have activated the "Elastic design" option when defining the basic parameters, the verification is performed in accordance with the elastic method as per equation 6.1.
- Stability verification as per EN 1993-1-1, equation 6.3.

The stability analyses of lateral buckling and lateral torsional buckling are based on the so-called model column method.

When applying the simplified analysis, an eigenvalue calculation is performed using the subspace method. The eigenvalue determination for the FE problem requires the solution of the general matrix eigenvalue problem for the smallest eigenvalue ηKi. This task is handled in *STS+* via the calculation module of the *BTII+* application The examination is performed for each load case combination and separately for each applicable design situation. This method ensures that the actually decisive failure situation in accordance with the safety concept can be determined.

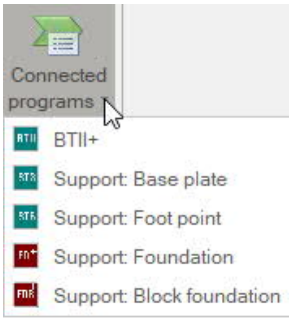
Verifications in the serviceability limit state

The displacements in direction of the different main axis and the resulting displacement are calculated in a first order analysis. The results are compared to the parameters defined by the user. The verification is considered successful when the calculated shifts are smaller or equal to the user-defined values.

Load transfer

Calling up further FRILO-designprograms - see also Interfaces.

The term load transfer refers basically to two extender functions, the transfer of the structural system to *BTII+* and the transfer of support reactions for the calculation of connected structures.



System transfer to the *BTII+* application

The first extender function is used for the export of the column system to the *BTII+* application for the calculation of more complex systems or to perform comparative calculations.

Higher requirements on the calculation of column systems which cannot be fulfilled by an application such as *STS+*, become relevant if the supporting conditions do not comply with the prescribed standard or if loads have to be included that produce planned torsion. Such systems cannot be verified using the model column method. They require second-order analyses with consideration of warping torsion. The *BTII+* application offers the necessary performance parameters for this task.

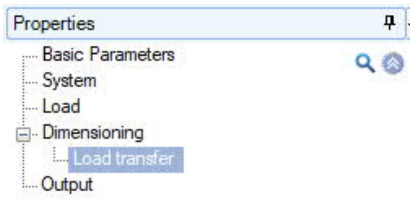
The column system is represented as a system section in the *BTII+* application. The supporting conditions correspond to the structural system of the column including the lateral supports.

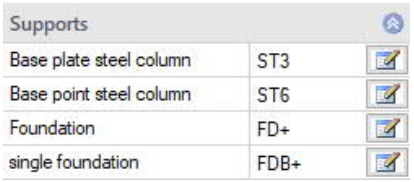
Transferring supporting forces

STS+ offers a load transfer feature to other applications for the calculation of connected structures and foundations.

An interface to the Isolated Foundation application *FD+* / *FDB+* allows the user to use the support reactions of the column system for the analysis of the foundations immediately underneath, if this is required. After selection of the appropriate foundation application it is launched automatically and the loading is generated in the form of the individual load cases used in *STS+*. The user must simply add the foundation-specific details and check the transferred load values.

The interfaces to *ST3* and *ST6* allow the transfer of the characteristic support reactions for the calculation of pinned or restrained column base structures.

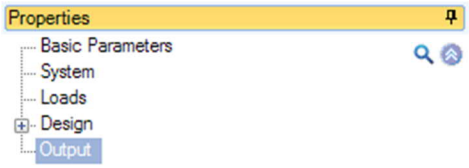




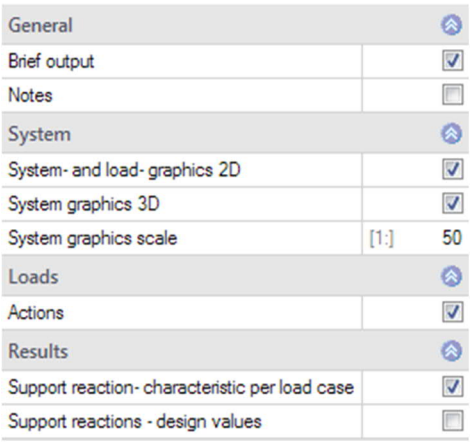
Output

By checking or unchecking the various output options, you can define the scope of the output (if an option is checked, the associated contents is integrated in the output document)

The options are described by tooltips and explanatory notes in the information section on the bottom of the screen.



Scale of system graph — by modifying the default scale you can adjust the size of the graph in the output document according to your requirements.

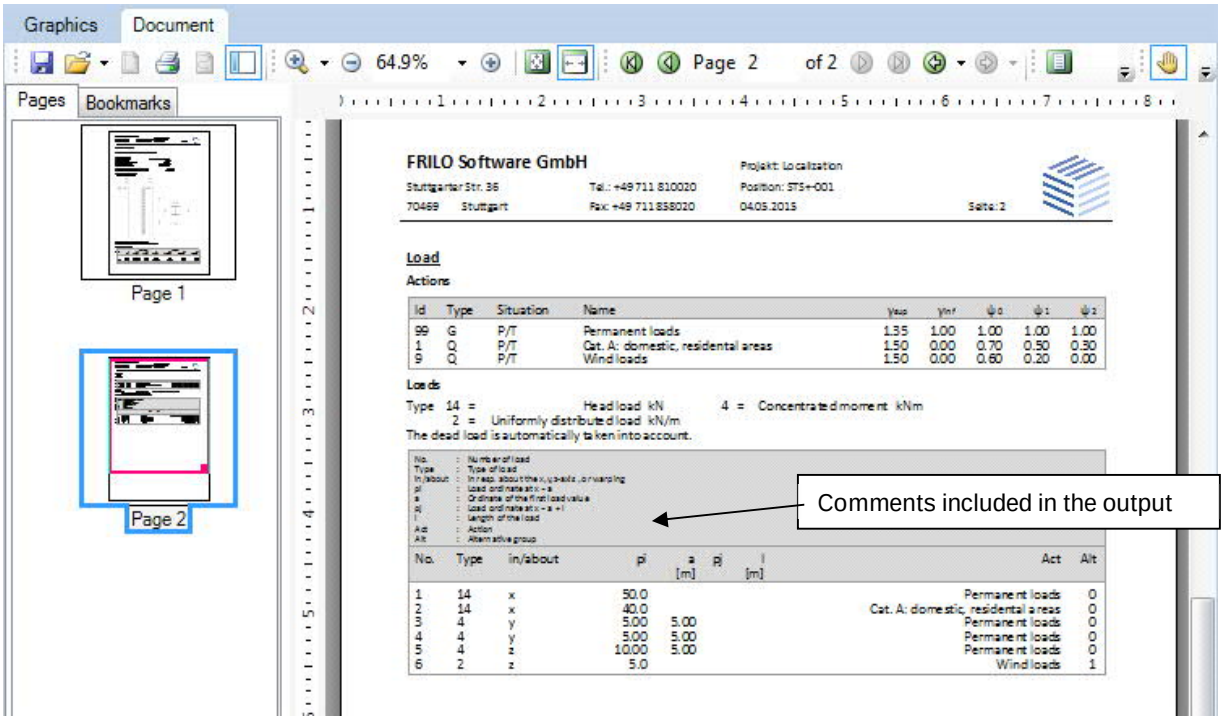


Output as a PDF file

The Document tab displays the document in PDF. You can display, save and print the PDF document.

A general description of the output options is available in the document:

Output and printing



Frequently asked questions

Structural system

Can I also calculate multi-span systems in *STS+*?

No. *STS+* provides for the calculation of single-span columns only. However, you can define lateral supports in the form of discrete or continuous supports. The application point relevant for the stability analyses can be defined either on the upper or the lower flange or in the shear centre.

Loads

Can I define loads that produce planned torsion?

No. Loads that produce planned torsion are not considered in *STS+*. The most important reason for this restriction is that the simplified model column analysis must not be used in a comparable load situation. In such a case, a second order analysis of torsional warping is required. We like to point out in this connection that our *BTII+* module is able to perform this task.

Calculation

Can I perform a second order analysis in addition to the verification based on the ideal column method?

No. Systems requiring second order analyses can be calculated with our *BTII+* module.