

Steel Chimney Design S8+

Table of contents

Application options	2
Basis of calculation	4
Verifications in the ultimate limit state	4
Verifications in the serviceability limit state	4
Load case combinations	4
Detail verifications	6
Flange connection	6
Base point	7
Flue opening	8
The wizard - fast entry of a basic systems	9
Basic parameters	10
Structural System	11
Segments	12
Cross-sections	14
Supports	15
Guying	15
Loading	16
Standard load cases	17
Additional load cases	17
Design	18
Output	19
Reference literature	21

Basic documentation - overview

In addition to the individual program manuals, you can find basic explanations on how to operate the programs on our homepage www.friilo.eu in the Download area ► Campus ► Manuals.

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

Application options

The S8+ program designs steel chimneys according to Eurocode 3, while taking into account the regulations of the National Annex. The program calculates the internal forces and the deformations according to the theory of elasticity in first-order and second-order analyses. The resonance frequencies used to perform the verifications of the structural safety, service strength and serviceability are also determined.

Relevant standards

The S8+ program performs structural safety analyses in accordance with EN 1993-1-1 and takes the corresponding National Annexes into account:

- DIN EN 1993-1-1/NA
- DIN EN 1993-3-2:2010:12 /NA

Snow and wind loads are based on the standards EN 1991-1-3 and EN 1991-1-4 and their National Annexes:

- DIN EN 1991-1-3/NA
- DIN EN 1991-1-4/NA

Fatigue

- DIN EN 1993-1-9/NA

Earthquake

- DIN EN 1998-1/NA
- DIN 4149

Wizard

The data necessary to define a simple system can be entered with a wizard. The basic system defined this way can be easily modified and supplemented subsequently via the interactive Graphical User Interface (GUI).

Structural systems

- Several length segments and different cross-sections
- Cantilevered chimneys restrained at the base
- Cantilevered chimneys supported at any point and restrained at the base or pinned
- Guyed chimneys with three or four ropes at several different guying points along the chimney (add-on licence S8-S required).

Loads

- Automatically generated standard load cases for wind and equivalent loads from transverse vibrations and from earthquakes
- User-defined additional load cases

Details

- Flange connection
- Base point
- Flue opening

Verifications

- Cross-sectional bearing capacity
- Serviceability
- Service strength
- Earthquake verification
- Verification of the flange connection
- Verification of the base point
- Verification of the flue opening

Calculation

The load cases and load case combinations are automatically generated from the defined effects of actions and the necessary verifications are performed. The decisive load case combination is determined for each limit state.

Add-on modules

The entry of guy ropes can be purchased as optional add-on S8-S.

Basis of calculation

Verifications in the ultimate limit state

The analyses in the ultimate limit state include the following 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) – no buckling resistance verification.
- Verification of the elastic cross-sectional bearing capacity (verification of the comparison stresses) as per EN 1993-1-1, eq. 6.1.

Verifications in the serviceability limit state

A second-order analysis is performed for the design situation selected in the basic parameters:

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

Load case combinations

The partial safety factors of the actions are included as per DIN EN 1993-3-1, Table NA.A.2.

Structural safety analysis for the fundamental combination

Case 1:

The following loads are combined in this load case:

- Permanent loads
- Pretensioning force/rope force
- Variable loads
- Wind Loads

Wind loads, thermal loads and loads from icing as well as other user-defined variable loads are thus considered. Forces due to transverse vibrations are not taken into account, however.

A difference is made between wind in the y-direction and wind in the x-direction as well as between wind 315° or 60°, depending on the arrangement of the guy ropes.

Case 2:

The following loads are combined in this load case:

- Permanent loads:
- Pretensioning force/rope force
- Equivalent loads from transverse vibrations
- Variable loads:

Equivalent loads from transverse vibrations: loads, thermal loads and loads from icing as well as other user-defined variable loads are considered this way. Wind loads in the incident flow direction are taken into

account, however. This verification is carried out for the fundamental frequency and, if necessary, higher resonant frequencies - as far as they can be excited by incident flow.

Structural safety analysis for the seismic combination

The following loads are combined in this load case:

- Permanent loads
- Pretensioning force/rope force
- Forces of inertia from seismic loads
- Variable loads

No wind loads are taken into account, neither as a result of direct incident flow nor as a result of transverse vibrations! The load case combination is only generated when the earthquake verification is enabled.

Service strength verification

The following loads are combined in this load case:

- Permanent loads
- Pretensioning force/rope force
- Equivalent loads from transverse vibrations

No additional variable loads are taken into account. Moreover, wind loads applying in the incident flow direction are not considered.

This verification is carried out for the fundamental frequency and, if necessary, higher resonant frequencies - as far as they can be excited by the incident flow.

Serviceability verification

The following loads are combined in this load case:

- Permanent loads
- Pretensioning force/rope force
- Variable loads
- Wind Loads

Wind loads, thermal loads and loads from icing as well as other user-defined variable loads are thus considered. Forces due to transverse vibrations are not taken into account, however.

A difference is made between wind in the y-direction and wind in the x-direction as well as between wind 315° or 60°, depending on the arrangement of the guy ropes.

Detail verifications

Flange connection

The design of the flange connections is performed in accordance with Petersen Stahlbau, 4th edition, p. 571 et seq. or p. 973 et seq.

In the design, the maximum tensile stress in the pipe jacket is evaluated. Using the bolt spacing, the affected width c per bolt is determined.

The program determines the compression springs of the flange "CD1" and the washer "CD2", as well as the tension spring of the screw "CS" and the torsion spring constant of the pipe "K".

This results in the total spring "C" and the distribution factors "p" and "q".

With the help of the tensile force "Z" and the pretensioning force of the screw " F_v ", the forces in the tension spring " F'_{vs} " and in the compression spring " F'_{vd} " are determined subsequently.

The existing tensile force is the maximum tensile stress in the pipe multiplied by the pipe thickness and the affected width c . The affected width results from the screw spacing referenced to the centre of gravity of the ring of screws.

An elasto-static verification is performed based on the L-model according to Petersen Stahlbau, 4th edition, p. 568. Taking into account the pretension, the program verifies at which tensile force " Z_{krit} " the connection will gape. In this verification, the tensile force "Z" is compared to the critical tensile force " Z_{krit} ".

Within the framework of the structural safety verification, the plasto-static verification of the L-model according to Petersen Stahlbau, 4th edition, p. 571 is performed. In this verification the three following failure mechanisms are examined:

- Bolt breakage
- Plastic hinge in section 2 (connection flange/pipe)
- Plastic hinge in section 3 (hole cross-section flange)

In addition, the program performs the verification of the weld seam between the flange and the pipe.

Base point

Anchor design:

To determine the tensile force of the anchors, the three options listed below are available:

- 1) Iteration as per Petersen Stahlbau, 4th edition, p. 986

In the iteration according to Petersen Stahlbau, the calculation of circular as well as annular base plates with screw anchors is performed using the equivalent bolt cross-section, which results from the ratio of the E-moduli of the anchor steel and the concrete. The result of the iterative calculation is compressive edge stress and tensile edge stress. The maximum tensile force of the anchor is calculated from the tensile edge stress.

- 2) Simple equation according to Petersen Stahlbau, 4th edition, p. 982, eq. (55) or p. 989

The maximum anchor force results in this verification from the following formula:

$$Z_0 = \frac{2}{n} \cdot \frac{M}{r_s} + \frac{N}{n}$$

r_s = radius of the ring of screws

- 3) Formula according to Stahlbau Handbuch, Volume 2, p. 1093, eq. 36.9-19

$$Z_0 = \frac{2}{n} \cdot \frac{M}{r_m} + \frac{N}{n}$$

r_m = mean radius of the pipe

Design of the base plate:

For the design of the base plate, the results (tensile force of the anchors Z_0) from the anchor design are used.

A cantilever arm, which is restrained at the pipe cross-section, is assumed as the structural system for the calculation of the base plate.

The cantilever moment, which results from the lever arm of the anchor spacing and the maximum tensile force, is calculated.

Furthermore, the moment is determined from the sheet bending in the flange according to eq. 36.7-9 [2].

The stress is determined based on the section modulus and the minimum of the moments from the base plate or the flange bending.

This stress is compared to the limit value of the stress σ_{Rd} .

If ribs are arranged, the design is performed for a plate restrained between the ribs according to Stahlbau Handbuch, Vol. 2, p. 1092.

In the calculation of the base plate, carding moments or the stiffness relationships between the pipe jacket and the base plate are not considered.

Rib design:

The design of the ribs is performed according to Kahlmeyer, Stahlbau section 8, p. 308 et seq.

The loading of the ribs results from the stress distribution in the pipe and the spacing of the ribs.

When ribs are arranged without an annular stiffener, the verification of a two-sided rib connection is performed. If an annular stiffener is arranged, the verification of a three-sided rib connection is performed.

Flue opening

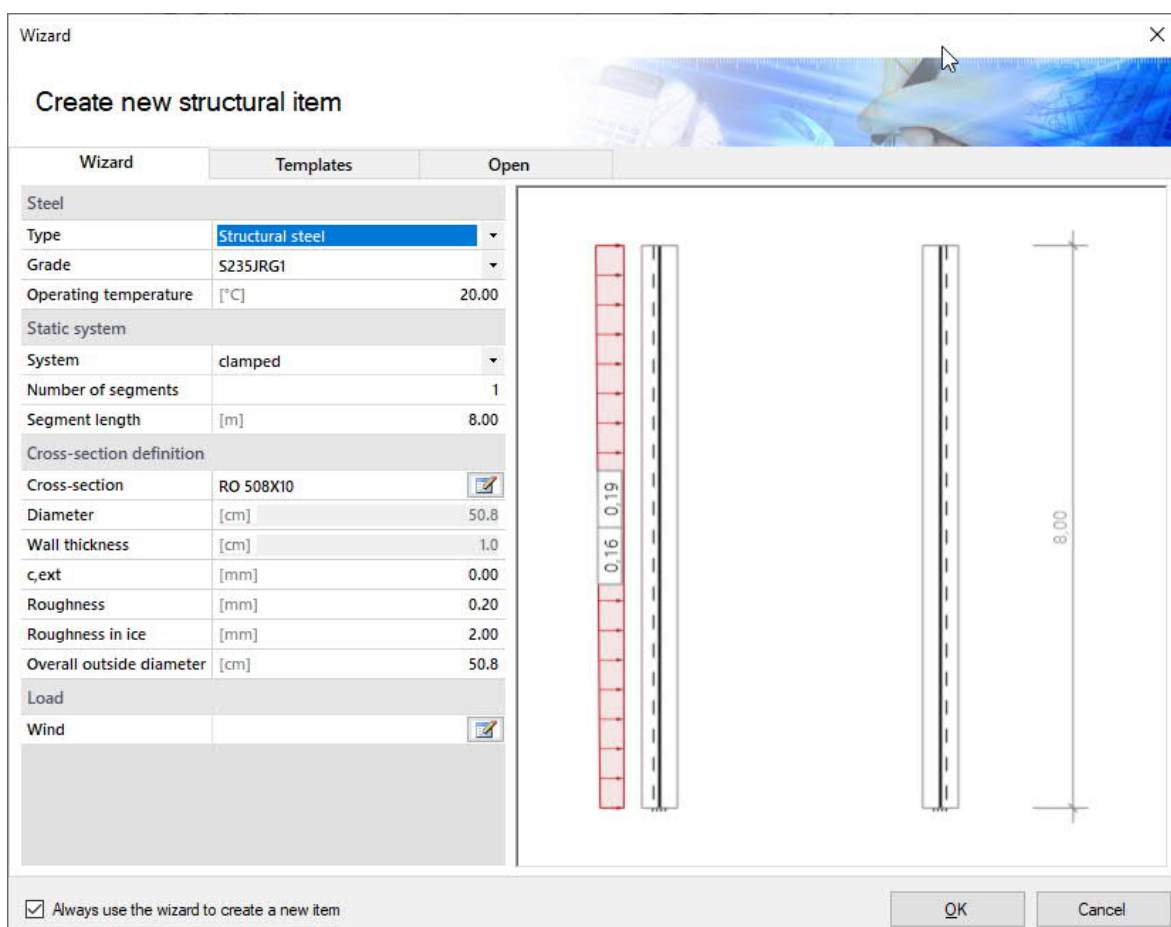
To verify the flue opening, the program performs a stress verification according to the bending theory in the weakened area. It is assumed that the longitudinal stiffeners of flat steel are arranged symmetrically on both sides of the flue opening.

The wizard - fast entry of a basic systems

The wizard is launched by default when you set up a new item. You can disable the wizard by unchecking the box at the bottom of the window. The entries required for the basic system can be made quickly and easily in the wizard.

They include:

- Material
- Structural system
- Definition of the cross-section
- Wind load



Wizard

Create new structural item

Wizard Templates Open

Steel

Type: Structural steel

Grade: S235JRG1

Operating temperature: [°C] 20.00

Static system

System: clamped

Number of segments: 1

Segment length: [m] 8.00

Cross-section definition

Cross-section: RO 508X10

Diameter: [cm] 50.8

Wall thickness: [cm] 1.0

c_ext: [mm] 0.00

Roughness: [mm] 0.20

Roughness in ice: [mm] 2.00

Overall outside diameter: [cm] 50.8

Load

Wind

☒ Always use the wizard to create a new item

OK Cancel

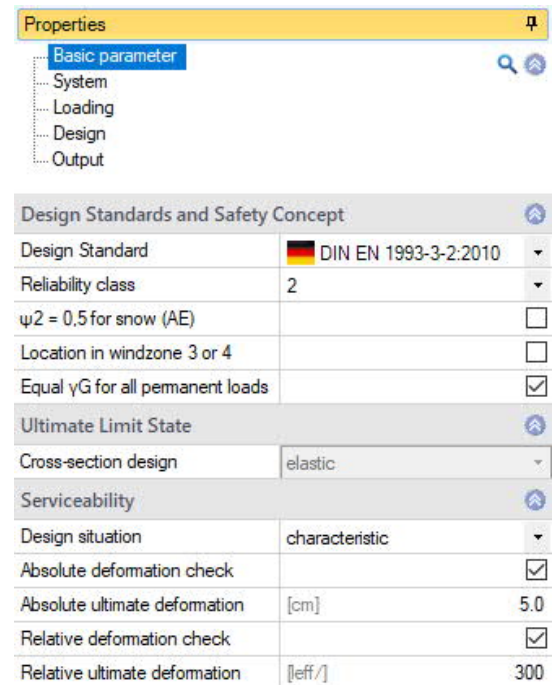
You can easily supplement and further edit this initially defined basic system subsequently.


See also the [Basic Operating Instructions-PLUS](#)

Basic parameters

Standard and safety concept

Design standard	Selection of the design standard and its National Annex.
Reliability class	Selection of the safety class to define the partial [safety] factors of the actions.
$\psi_2 = 0.5$ for snow	You can select whether the combination coefficient ψ_2 in the seismic design situation (AE) should be raised to the value 0.5 for the snow action. (See introductory decrees of the German federal states, e. g. Baden-Württemberg).
Location in wind zone ...	Check this option, if the building is located in wind zone 3 or 4. In this case, you need not consider snow as an accompanying action to wind, which is the leading action.
Equal γ_G for ...	When you check this option, all permanent loads and/or load cases are included with the same partial safety factor ($\gamma_{G,sup}$ or $\gamma_{G,inf}$). Otherwise, all permanent loads and/or load cases are combined with each other using $\gamma_{G,sup}$ and $\gamma_{G,inf}$.



Design Standards and Safety Concept	
Design Standard	 DIN EN 1993-3-2:2010
Reliability class	2
$\psi_2 = 0.5$ 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	elastic
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	[l _{eff} /] 300

Structural safety

The cross-section is designed in an elastic analysis as per DIN EN 1993-1-1, eq. 6.1

Serviceability


Design situation	Select the design situation the verifications in the serviceability limit state should be based on (characteristic, frequent and quasi-permanent).
Absolute deformation check	Performs the serviceability verification with consideration of the difference in deformation in comparison to the undeformed system.
Absolute ultimate deformation	Indicates the permitted maximum relative deformation of the structural system.
Relative deformation check	Performs the serviceability verification with regard to the effective lengths, which are determined by the inflection points (moment passage) of the bending line.
Relative ultimate deformation	Indicates the permitted maximum relative deformation of the structural system.

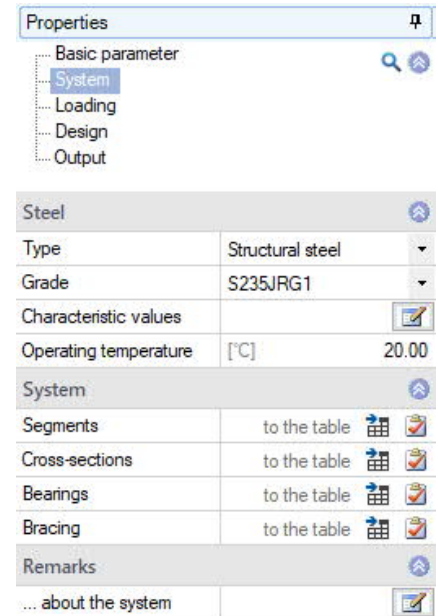
Structural System











Steel material

You can select the standardised steel type and steel grade or the user-defined entry - in this case you manually can enter the characteristic values. The operating temperature should be specified in addition. Depending on the operating temperature, the program automatically determines the modulus of elasticity, the yield strength f_{yk} and the specific weight γ .

- Structural steel
- Weathering steel
- Fine-grained steel
- Stainless steel
- High-temperature steel
- User-defined steel:

Click on the  icon to display the dialog for the definition of the steel parameters.



Properties	
Basic parameter	
System	
Loading	
Design	
Output	
Steel	
Type	Structural steel
Grade	S235JRG1
Characteristic values	
Operating temperature	[°C] 20.00
System	
Segments	to the table  
Cross-sections	to the table  
Bearings	to the table  
Bracing	to the table  
Remarks	
... about the system	

Structural System

The tabs below the graphic provide access to the tables for the data entry.

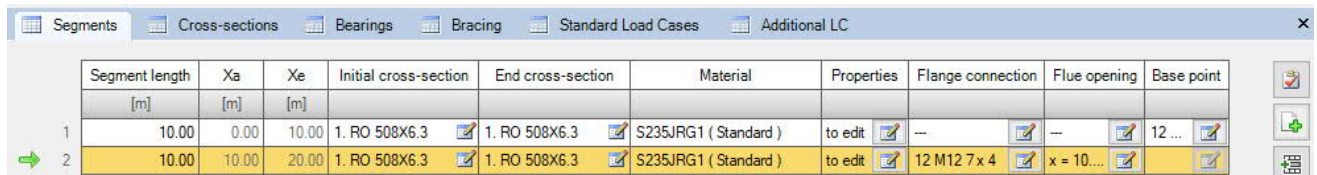
- [Segments](#)
- [Cross-sections](#)
- [Supports](#)
- [Bracing](#)

Remarks

You can enter [comments](#) on the system, which are included in the output document. You can optionally hide them in the output.

Segments

Define the length of the individual segments and their start and end cross-sections as well as the material - see also [Cross-section selection](#).

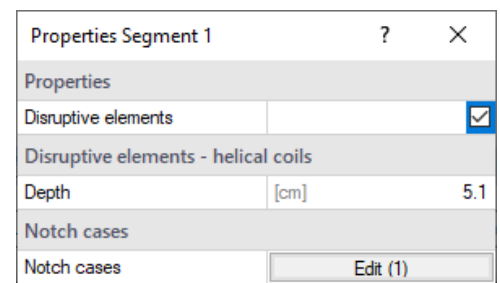


	Segment length [m]	Xa [m]	Xe [m]	Initial cross-section	End cross-section	Material	Properties	Flange connection	Flue opening	Base point
1	10.00	0.00	10.00	1. RO 508X6.3	1. RO 508X6.3	S235JRG1 (Standard)	to edit	---	---	12 ...
2	10.00	10.00	20.00	1. RO 508X6.3	1. RO 508X6.3	S235JRG1 (Standard)	to edit	12 M12 7 x 4	x = 10...	

For each length segment, you can define properties and a flange connection, if there is more than one segment.

Properties

Disruptive elements Optional enabling of disruptive elements as an aerodynamic measure. Entry of the depth of helical coils according to DIN EN 1993-3-2, Annex B, eq. (B.2b) in the range $t \leq 0.1 \cdot d$ to $0.12 \cdot d$. Due to the application of the aerodynamic measure, the basic value of the aerodynamic excitation force coefficient $c_{lat,0}$ according to DIN EN 1993-3-1, NA.B.4.5.2, figure NA.B.8 is automatically reduced. The effective length l_w is assumed to be the total height of the segment. If $l_w/h < 0$, the program automatically sets the value $c_{lat} = 0.1$. According to DIN EN 1991-1-4, E.1.5.2.1, the effective length factor for calculating the transverse vibration amplitude is $K_w = 1$.



Properties Segment 1	
Properties	
Disruptive elements	<input checked="" type="checkbox"/>
Disruptive elements - helical coils	
Depth	[cm] 5.1
Notch cases	
Notch cases	Edit (1)

Note: According to DIN EN 1991-1-4, Annex E, the arrangement of helical coils as an aerodynamic measure against vortex-excited vibrations may only be applied with a Scruton number greater than 8.

Notch cases Click on the "edit" button. In the Notch case dialog, you can add new notch cases by clicking on the +-icon (adds a new row):
Segment coordinate: Determines the verification point for this notch case.










The verification can be based on the circumference or an angle to the pipe.

The "Edit" button to the right of the notch case selection displays additional details of the notch case.

Flange connection

A prerequisite for arranging a flange connection is that the chimney has at least two segments. The flange connection is defined for each segment. The flange can be external or internal. The user must define the thickness, the width, the vertical projection on the pipe as well as the weld seam thicknesses between the flange and the supporting pipe on the upper or lower pipe cross-section in the data-entry mask. Based on the specified number of anchors these are evenly distributed circumferentially around the flange. The anchor spacing to be defined refers to the distance between the outer edge of the flange and the axis of the anchor.







In order to remove a flange connection that has already been defined, you have to uncheck "Flange connection available" in the menu window for defining the flange connection.

Definition of the flange connection		
<div>  </div>		
Flange connection available		<input checked="" type="checkbox"/>
Darstellung	2D graphic top view	
<div>  </div>		
Tubular profile RO 508X6.3		
<div>  </div>		
Tubular profile RO 508X6.3		
<div>  </div>		
Ring flange		
Position	outside	
Thickness	[cm]	4.0
Width	[cm]	7.0
Overhang	[cm]	0.5
awu	[cm]	0.4
awo	[cm]	0.4
<div>  </div>		
Bolts		
Number		12
Distance	[cm]	4.0
Bolt	M12 - 10.9 (Black bolt)	
<div>  </div>		
Steel material		
Name		S235JRG1
Temperature		20.0

Flue opening

When defining a flue opening, the distance from the lower edge of the segment must be specified to define the position. The opening angle must be defined in addition. To enter the stiffeners, you must specify the width, the thickness, the angle of the stiffener to the pipe centreline and the outer centre of the stiffener.

In order to remove a flue opening that has already been defined, you have to uncheck "Flue opening exists" in the menu window for defining the flue opening.

Definition of the opening		
<div>  </div>		
Mast opening present		<input checked="" type="checkbox"/>
Distance from bottom edge section	[m]	0.00
Pipe diameter at X	[cm]	50.8
<div>  </div>		
Tubular profile RO 508X6.3		
<div>  </div>		
Tubular profile RO 508X6.3		
<div>  </div>		
Geometry		
Opening angle	α [Grad]	45.00
Width	[cm]	7.1
<div>  </div>		
Stiff		
Width	[cm]	4.0
Thickness	[cm]	1.0
Angle	β [Grad]	45.00
Imperfection	[cm]	1.0
<div>  </div>		
Steel material		
Name		S235JRG1
Temperature		20.0

Base point

The base can be defined as a circular or annular base plate.

For a circular base plate, the user must define the thickness and the diameter of the base plate in the data-entry mask. For an annular base, the outside and inside diameters must be specified.

There are three different options for determining the tensile force of the anchors, which you can select from the drop-down menu. Furthermore, you have to define the number of bolts and the distance from the outer edge of the flange to the axis of the bolts.

When selecting the anchorage, you can choose metric bolts or enter user-defined bolt parameters. Based on the specified number of anchors these are evenly distributed circumferentially around the flange.

Moreover, you have the option to arrange ribs. Based on the specified number of anchors, the ribs are evenly distributed over the pipe cross-section. In addition to ribs, you can define an angular stiffener.

Definition of the base point	
with base	<input checked="" type="checkbox"/>
Presentation	2D graphic top view
Tubular profile RO 508X6.3	
Footplate	
Shape	circular
Thickness	[cm] 4.0
diameter	[cm] 70.0
Design anchor	
Determination of tensile force	Iteration
E-modulus ratio	7.00
Number	12
Distance	[cm] 4.0
Anchor	12 M20 70 x 4
Ribs	no
Steel material	
Name	S235JRG1
Temperature	20.0

In order to remove a base that has already been defined, you have to uncheck "With base" in the menu window for defining the base point.

Cross-sections

Segments Cross-sections Bearings Bracing Standard Load Cases Additional LC							
	Cross-section	Diameter	Wall thickness	c,ext	Roughness	Roughness in ice	Overall outside diameter
		[cm]	[cm]	[mm]	[mm]	[mm]	[cm]
1	RO 508X6.3	50.8	0.6	0.00	0.20	2.00	50.8
2	RO 508X6.3	50.8	0.6	0.00	0.20	2.00	50.8

You can set up the desired cross-sections on the "Cross-sections" tab. Several pipe cross-sections are available as standard cross-sections for selection. Moreover, you have the possibility of customising a standard cross-section to create a user-defined cross-section. Furthermore, you can enter an equivalent roughness of the supporting pipe according to EN 1991, section 7.9.2, table 7.13, a corrosion allowance as well as an equivalent roughness for ice. By entering the total outer diameter, insulation or cladding that affects the inclusion of wind load can be taken into account.

See also the document [Selection of Cross-Sections in the PLUS Programs](#).

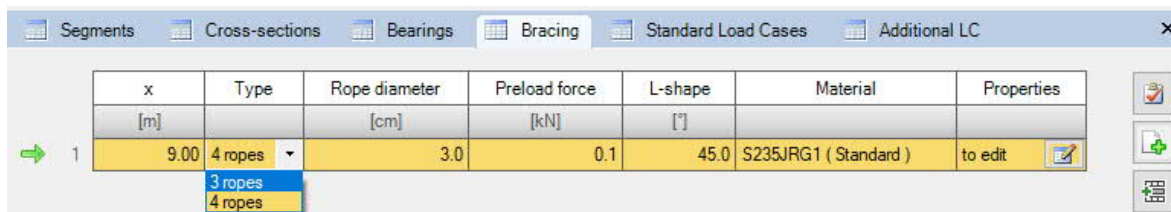
Supports



	x	ux	uy	uz	phix	phiy	phiz
	[m]	[kN/m]	[kN/m]	[kN/m]	[kNm/rad]	[kNm/rad]	[kNm/rad]
1	0.00	rigid <input checked="" type="checkbox"/>	rigid <input checked="" type="checkbox"/>	rigid <input checked="" type="checkbox"/>	rigid <input checked="" type="checkbox"/>	rigid <input checked="" type="checkbox"/>	rigid <input checked="" type="checkbox"/>

On the "Bearings" tab, you can define supports at the base point or additional restraints on the system. Six degrees of freedom are available for selection. The support can be defined as either rigid, articulated or elastic, or by specifying a spring value.

Guying



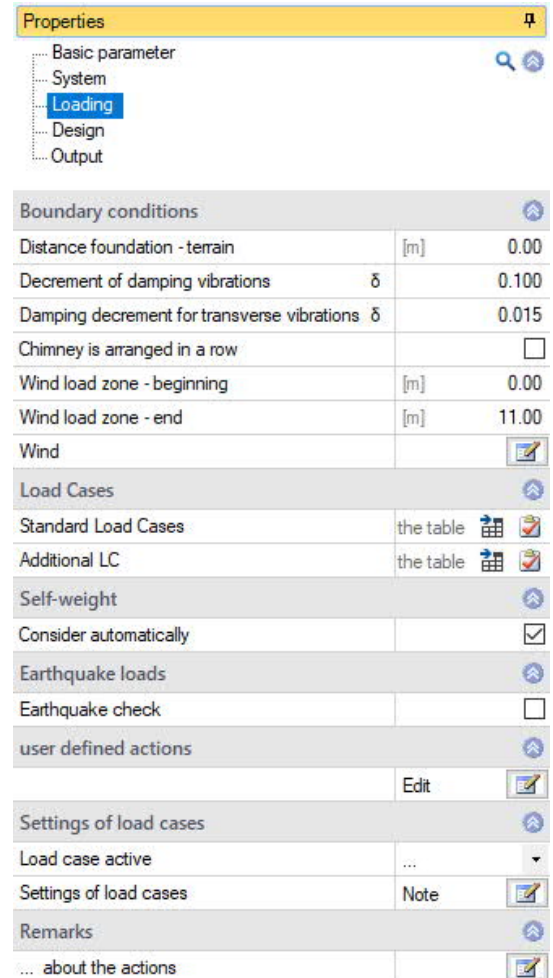
	x	Type	Rope diameter	Preload force	L-shape	Material	Properties
	[m]		[cm]	[kN]	[°]		
1	9.00	4 ropes	3.0	0.1	45.0	S235JRG1 (Standard)	to edit

On the "Bracing" tab, you can define guy ropes on the system if the [Add-on S8-S](#) was licensed. You can select the height of the guying on the system as well as the number of ropes. Three or four ropes are available for selection. Moreover, you can apply a pretensioning force to the guy ropes. The pretensioning force must always be equal to or greater 0.1 kN. Furthermore, the angle and the rope material must be defined. In the table column "Properties", the height of the individual anchorage points of the ropes (only for 4 ropes) as well as a possible ice hanging or also a self-defined base area of the guy ropes (only for 4 ropes) can be specified. The thickness of the ice layer around the guy rope is only applied in combination with a load case involving action of icing. You can enable the automatic inclusion of wind action. In this case, wind action is automatically applied to all guy ropes.

Loading

Border conditions

Distance foundation - terrain	Defines the distance from the upper edge of the foundation to the top edge of the ground (downwards negative)
Damping decrement ... δ	Logarithmic damping decrement for vibrations in the wind direction or for transverse vibrations.
Chimney arranged in a row	If you check this option, you can define an arrangement of chimneys in a row. You need to specify the chimney spacing. The increase factor K for circular cylinders arranged in a row as per DIN EN 1991-1-4, table 7.14 is displayed.
Wind load zone ...	Defines the start and end position x of the wind application area on the supporting pipe.
Wind	Click on the edit button, to display the dialog for the selection of the municipality and the wind parameters.



Properties

- Basic parameter
- System
- Loading**
- Design
- Output

Boundary conditions

Distance foundation - terrain	[m]	0.00
Decrement of damping vibrations δ		0.100
Damping decrement for transverse vibrations δ		0.015
Chimney is arranged in a row		<input type="checkbox"/>
Wind load zone - beginning	[m]	0.00
Wind load zone - end	[m]	11.00
Wind		

Load Cases

Standard Load Cases	the table	
Additional LC	the table	

Self-weight

Consider automatically	<input checked="" type="checkbox"/>
------------------------	-------------------------------------

Earthquake loads

Earthquake check	<input type="checkbox"/>
------------------	--------------------------

user defined actions

	Edit	
--	------	--

Settings of load cases

Load case active	...	
Settings of load cases	Note	

Remarks

... about the actions	
-----------------------	--

Load cases

The data-entry tables for standard and additional load cases are accessible via the tabs below the graphic screen.

You can display further information on standard/additional load cases by selecting "Settings on load cases" and "Note".

See [Standard load cases](#)

See [Additional load cases](#)

Self-weight

Check this option to include the self-weight automatically as a permanent load in the calculation.

Earthquake loads

When you check this option, the earthquake loads are taken into account when creating the load cases and superpositions. The dialog "Basic values for the determination of the ground acceleration response spectrum" is displayed.

See also the description in the program [GEO](#).

You can select the design spectrum (plateau value/resonant frequency).

User-defined actions

In addition to the pre-defined actions, that are available for selection in the load table, you can also specify user-defined actions. These are added to the load table for selection.

Settings of load cases

Load case active The drop-down list allows you to enable or disable load cases. Disabled load cases are not considered in the calculation.

Settings of load cases A note concerning the control of load cases is displayed.

Remarks

You can enter [comments](#) on the loading, which are included in the output document.

Standard load cases

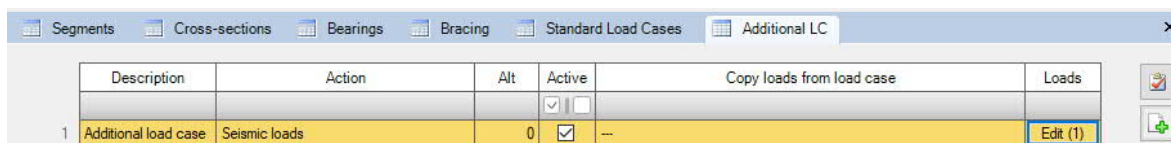
The load cases Wind load and Equivalent load (from transverse vibrations or earthquake) generated by the program are standard load cases. They are generated automatically on the basis of the chimney dimensions and the wind zone. You cannot edit them and they are regenerated each time you change the structural system. By removing the tick from the Active checkbox, you can disable the associated load case. Click on "Details" to display the corresponding values.



	Description	Action	Alt	Active	Loads
1	Generated wind loads	Wind loads	111	<input checked="" type="checkbox"/>	Details (1)
2	Ersatzkräfte (Querschwingungen):1. (6)	Querschwingungen	151	<input checked="" type="checkbox"/>	Details (43)
3	Ersatzkräfte (Querschwingungen):2. (1)	Querschwingungen	151	<input checked="" type="checkbox"/>	Details (43)

Additional load cases

In addition, you can enter user-defined load cases, so-called additional load cases. For example, a load due to icing of the supporting pipe, an additional wind load or also a thermal load can be taken into account. The action generating the load can be selected by clicking on the arrow in the column "Action". By clicking on "Edit" in the "Loads" column, you can define the load value.



	Description	Action	Alt	Active	Copy loads from load case	Loads
1	Additional load case	Seismic loads	0	<input checked="" type="checkbox"/>	---	Edit (1)

Design

Remarks

You can enter comments on the design, which are included in the output document.

Output

Start the calculation by activating the corresponding button in the upper menu bar. You can display the result graphics and the verifications.

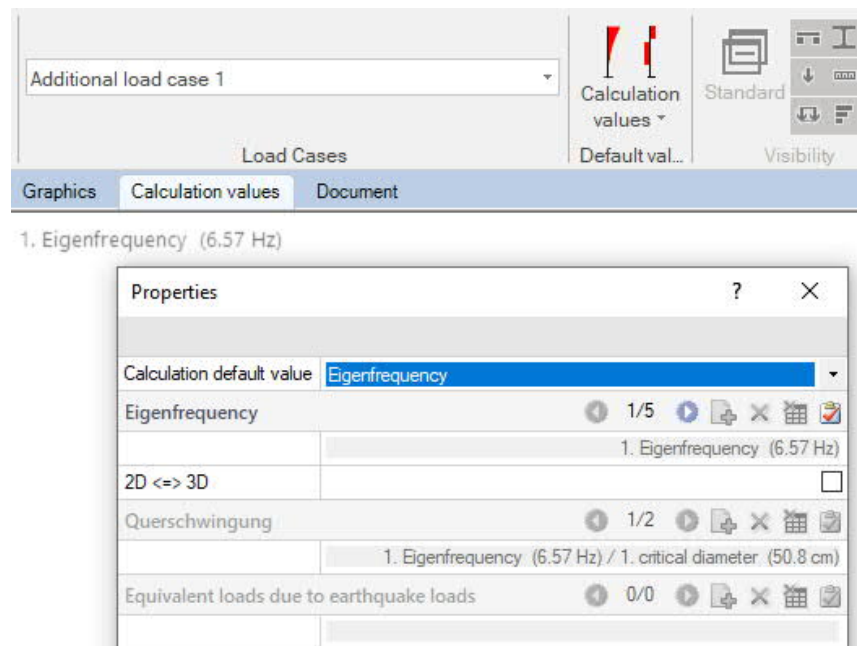
Utilization is displayed on bottom right in the graphic window after the calculation has been performed.

View selection

The tabs "Graphics", "Calculation Values" and "Document" allow you to toggle between the representation of the structural system and the preview of the text documents.

Calculation values

Click on the lower area of the button "Calculation values" in the upper menu bar to access a properties dialog and make your selections: Resonance frequency, equivalent loads due to transverse vibrations and due to seismic loads.

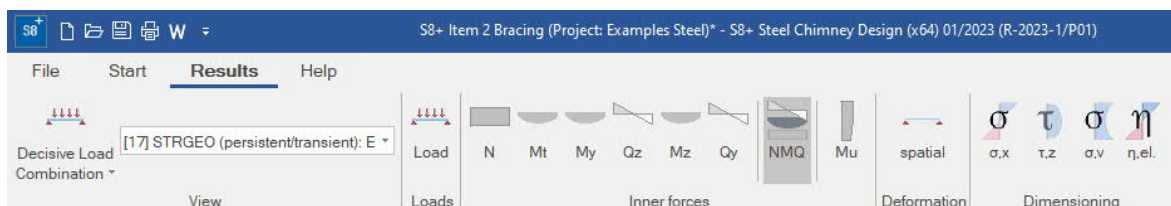


Visibility

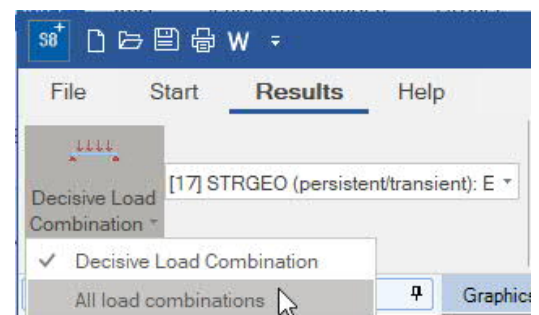
Via the button in the upper toolbar, you can enable or disable individual representations in the graphic.

Results

You can display the result graphics via the "Results" tab or via the "Results" icon in the ribbon bar.



Furthermore, you can display not only the results of the decisive load cases, but also all calculated load cases.



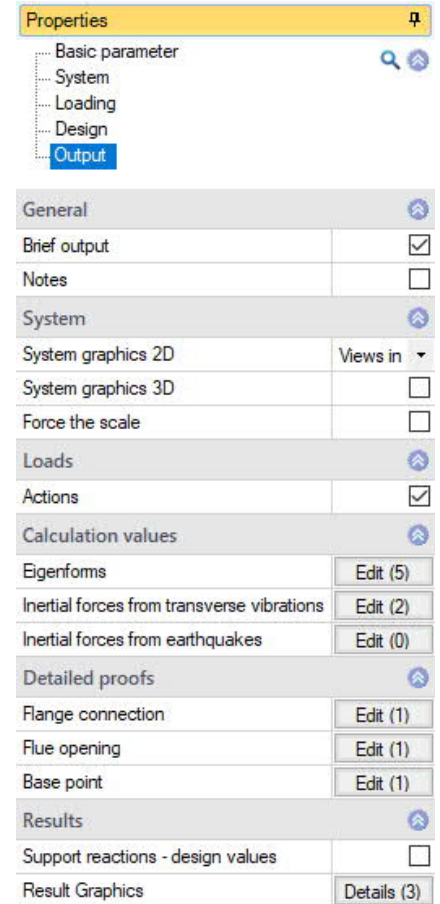
Output scope

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

Output as a PDF document

Via the [Document](#) tab, you can display the document in PDF and print it.

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



The screenshot shows the 'Properties' dialog box with the 'Output' tab selected. The dialog is organized into several sections, each with a header and a set of options. The 'Output' tab is highlighted in blue. The sections and their options are as follows:

Section	Option	Status
Basic parameter	System	<input type="checkbox"/>
	Loading	<input type="checkbox"/>
	Design	<input type="checkbox"/>
	Output	<input checked="" type="checkbox"/>
General	Brief output	<input checked="" type="checkbox"/>
	Notes	<input type="checkbox"/>
System	System graphics 2D	Views in ▾
	System graphics 3D	<input type="checkbox"/>
	Force the scale	<input type="checkbox"/>
	Loads	<input checked="" type="checkbox"/>
Calculation values	Eigenforms	Edit (5)
	Inertial forces from transverse vibrations	Edit (2)
	Inertial forces from earthquakes	Edit (0)
Detailed proofs	Flange connection	Edit (1)
	Flue opening	Edit (1)
	Base point	Edit (1)
Results	Support reactions - design values	<input type="checkbox"/>
	Result Graphics	Details (3)

Reference literature

- [1] Petersen Stahlbau, 4th fully revised and updated edition
- [2] Stahlbau Handbuch, Volume 2, second revised edition, Stahlbau-Verlagsgesellschaft mbH. Cologne 1985
- [3] Kahlmeyer, Stahlbau