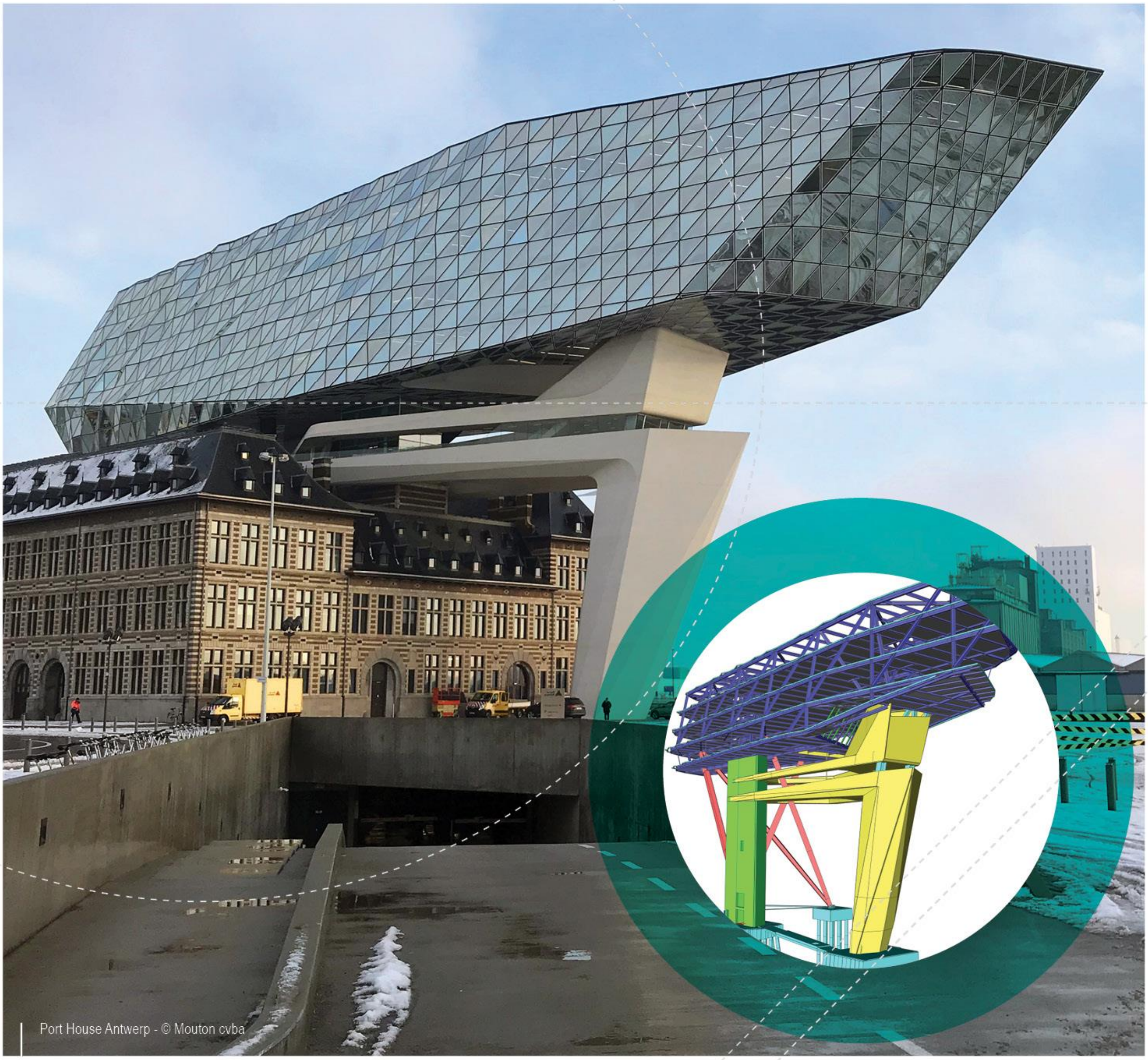
SCIA ENGINEER



TOPIC TRAINING

BUCKLING LENGTHS FOR STEEL

Table of Contents

Introduction

This course will explain the principles and the use of system lengths in SCIA Engineer. The buckling parameters and the related buckling lengths will be discussed, in combination with the relative deformation of 1D elements.

Most of the options in the course can be calculated in SCIA Engineer with the **Concept edition**.

For some functionality an extra module (or edition) is required, but this will always be indicated in those paragraphs.

Buckling factors and lengths

Since SCIA Engineer 18.0 a new dialog is introduced for applying buckling settings on a specific buckling system called **System lengths and buckling settings**. Prior to SCIA Engineer 18.0 there was a dialog for the buckling settings called "buckling and relative lengths" which offered similar settings but without the graphical window and even without the results.

The new dialog is an improvement of buckling and relative lengths in order to offer the user a better user interface and also a better user experience. The settings have been grouped in a more logical order so that the user can navigate seamlessly within the dialog.

Automatic placement of buckling supports

SCIA Engineer will automatically recognize every nodal support, beam connection, for the placements of buckling supports.

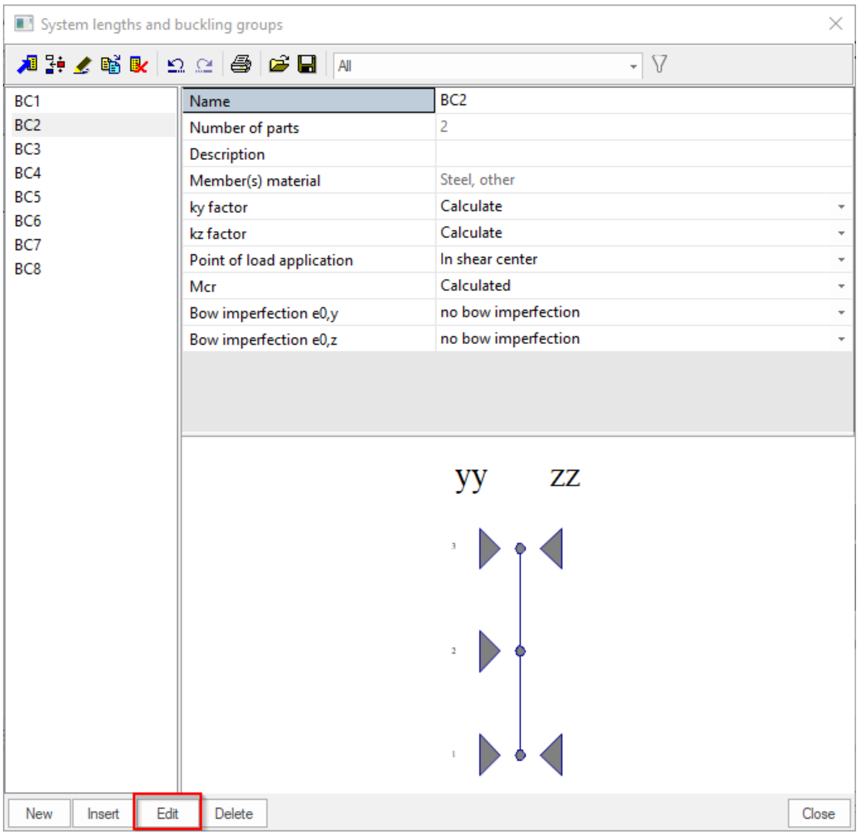
Example: Buckling supports.esa

In this example, several beams are connected to the columns. A buckling support is automatically placed at every location of a connection between the members.

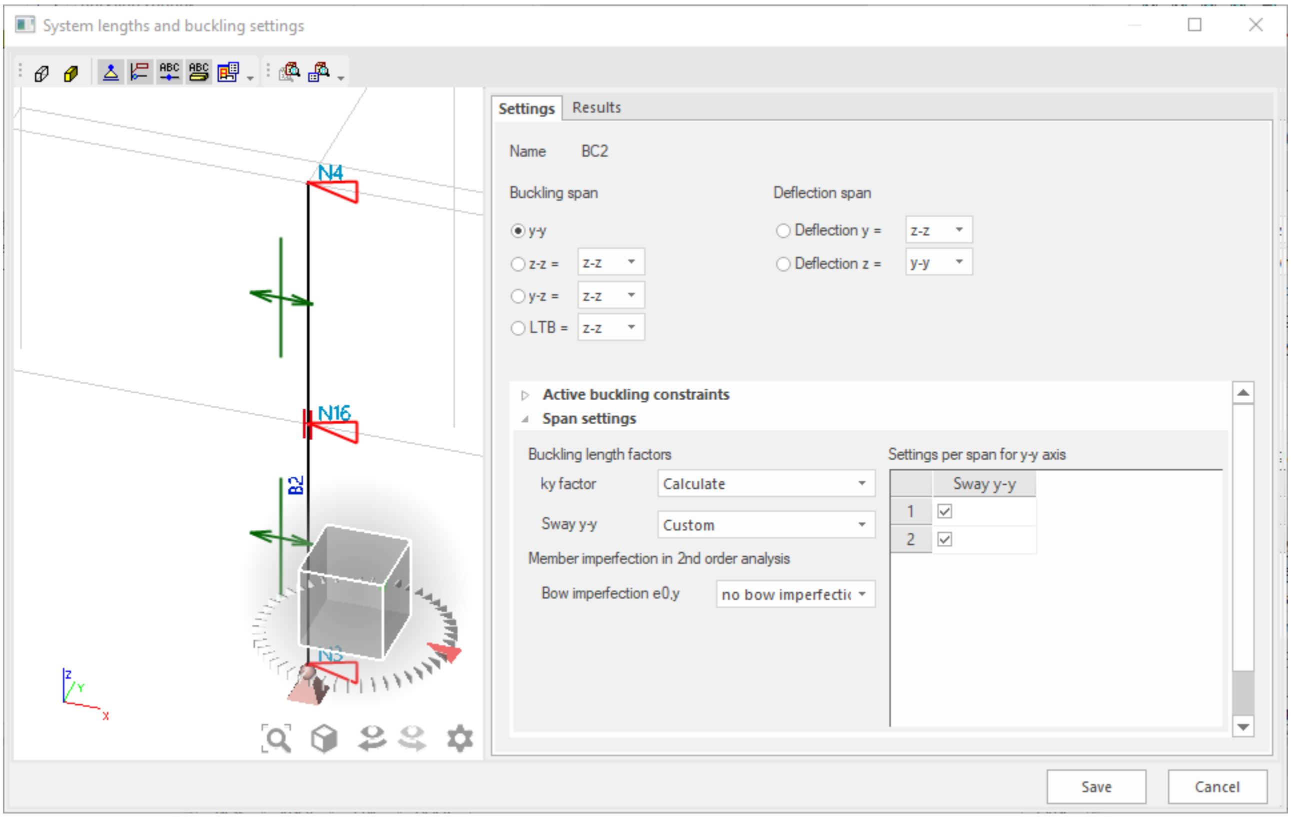
Note: Buckling supports can only be placed in structural nodes.

Consider column B2. This column has 3 nodes: N3, N16 and N4. The local X direction goes from the bottom to the top of the column, so internal for SCIA Engineer N3 is the first node of this column and N4 is the last one.

In the properties window for this buckling group (System lengths and buckling groups), the user can check properties of this particular buckling group. With the option ‘Edit’, buckling system of the element can be viewed and changed.



Default the following option will appear:



System lengths and buckling settings can also be accessed either:
- via Libraries > Structure, analysis > System lengths and buckling groups > click on new for creating a new buckling group or click on edit to modify an existing buckling group.
- via 1D-member property > System lengths and buckling settings.

Explanation of the system lengths:

- The first node (according to the local x-axis) is node N3, the last one is node N4.
- yy direction:
  - This means **around** the local y-axis. So the column will deform in the local z-direction.
  - Around the y-axis, node N3 is supported. In node N16 a horizontal beam in the local z-direction can be found and the column will be supported around the local y-axis (yy) in node N16. Also in node N4 a horizontal beam in the local z-direction can be found and the column will also be supported around the local y-axis (yy) in node N4. This is indicated with the triangles in this window:
    - Supported in node N3
    - Supported in node N16
    - Supported in node N4
- zz directions:
  - This means **around** the local z-axis. So the column will deform in the y-direction.
  - Around the z-axis, node N3 is supported. In node N16 no beam can be found in the local y-direction in this point, thus column B2 is not supported around the z-axis in node N16. In node N4 a horizontal beam in the local y-direction can be found and the column will be supported around the local z-axis (zz) in node N4. This is indicated with the triangles in this window:
    - Supported in node N3
    - Not supported in node N16
    - Supported in node N4
- The system length will be taken as follows:

- Around the y-axis: the length between node N3 and node N16 for the first part of the column (2,5m) and the length between node N16 and node N4 for the second part of the column (2,5m).
- Around the z-axis: the length between node N3 and node N4, so 5m.
- This can also be found in the menu 'Steel > Beams > Steel slenderness':

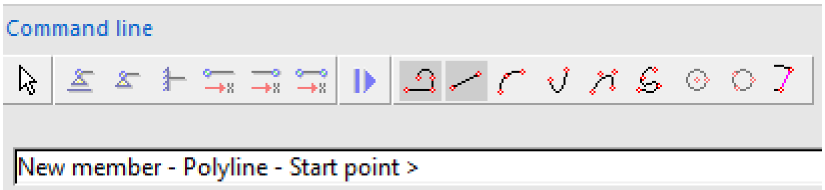
Steel slenderness

Linear calculation

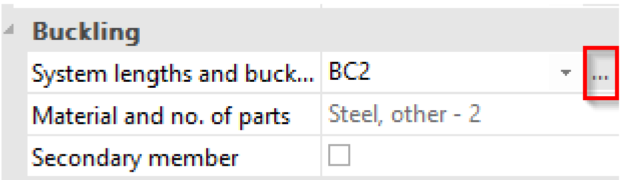
Member	CS Name	Part	Sway y Sway z	Ly [m] Lz [m]	ky [-] kz [-]	ly [m] lz [m]	Lam y [-] Lam z [-]	lyz [m]	l LTB [m]
B2	CS1	1	Yes	2,500	3,98	9,954	78,22	5,000	5,000
			No	5,000	0,82	4,094	54,78		
B2	CS1	2	Yes	2,500	3,00	7,506	58,98	5,000	5,000
			No	5,000	0,82	4,094	54,78		

- In this window, the user can easily check the system length (L_y and L_z), the buckling factors (k_y and k_z) and also the buckling length ($l_y = k_y \times L_y$ and $l_z = k_z \times L_z$)

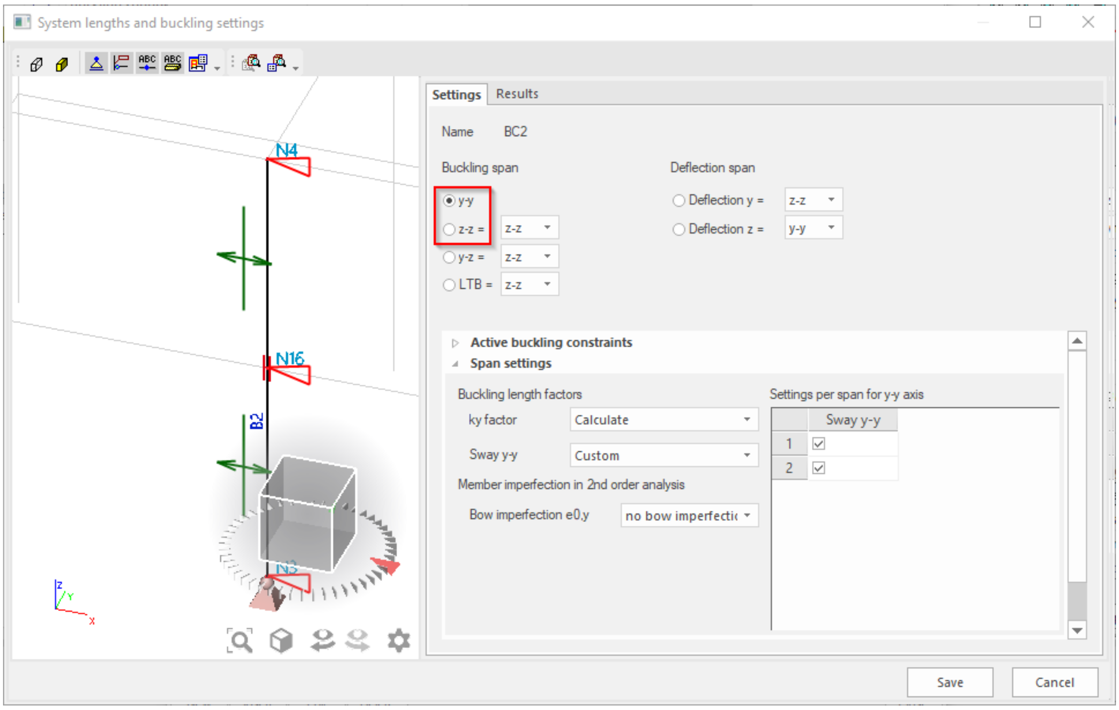
- Compare column B2 and columns B18 and B19 with each other. They should have exactly the same system lengths. The only difference between those columns is that column B2 was inputted as one element of 5m and columns B18+B19 are divided into two parts of 2,5m. SCIA Engineer will consider those two columns also as one buckling system:
  - If the local axis are exactly in the same direction.
  - If no hinges have been inputted between the two columns.
    - **Note**: If members are inserted as a **Polyline**, the direction of the members may be different. The complete polyline is seen as one buckling system.



- There is also a graphical representation of the system lengths.
  - Select column B2.
  - Go to the properties menu and click on **System lengths and buckling settings**:



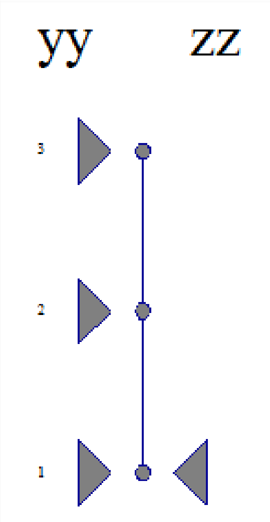
- Now the user can choose between the yy and the zz axis:



Manual placement of buckling supports

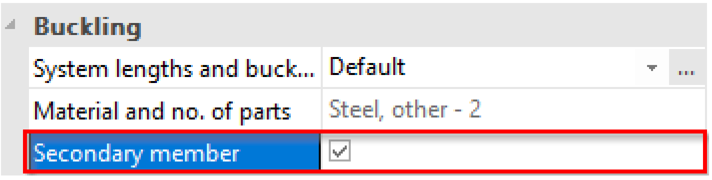
Since a buckling support (triangle) is placed at every structural node where there is a nodal support or a connecting member, it can be necessary to manually change this configuration.

Select column B3 and open the System lengths and buckling groups window to have a look at the buckling system. The system lengths are as follows (as expected, because there is a horizontal beam in the local z-direction on each node):



When looking at the rendered view of the structure, it is clear that beam B17 is too weak to have an influence on the system length of column B3. In SCIA Engineer, there is a possibility to exclude a beam from a buckling system. This can be done by inputting **secondary beam properties** to the element.

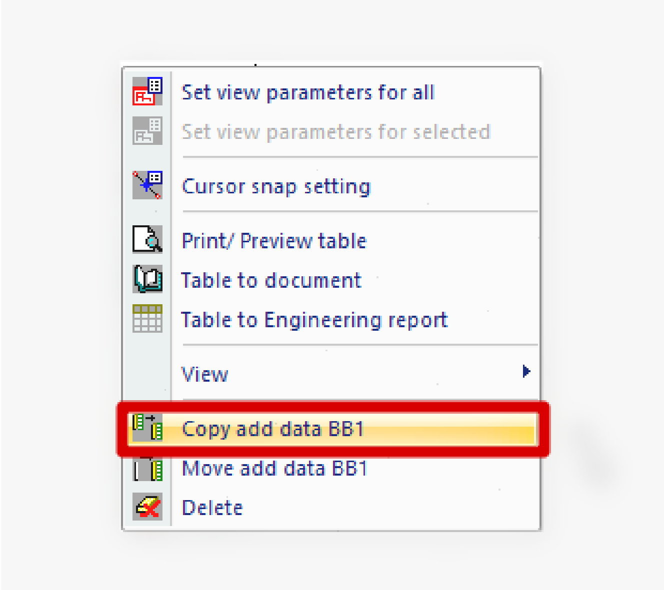
So select beam B17 and go to the properties menu of this 1D member:



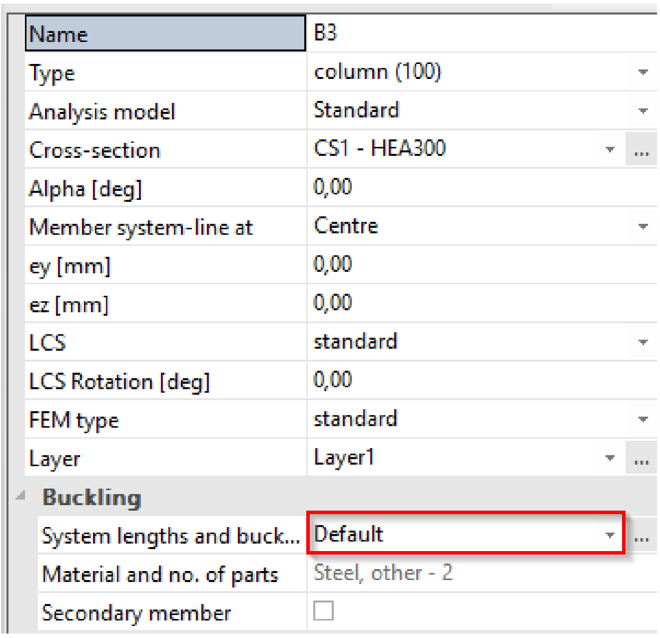
It is possible to indicate that beam B17 is a secondary beam and should not be taken into account in the system lengths:

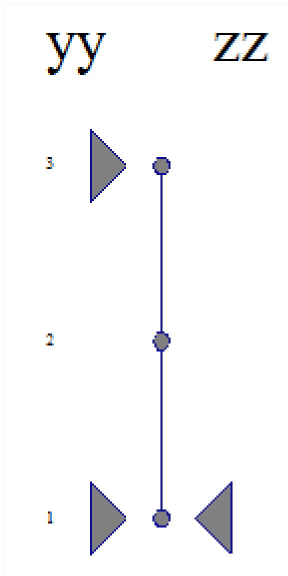
Afterwards, a label will be displayed on beam B17, indicating that this beam consists of some buckling data. This buckling data is **Additional data** and can be copied to other members in the structure.

This can be done by selecting the label, click with the right mouse button in the screen and choose the option **Copy add data BB1**.



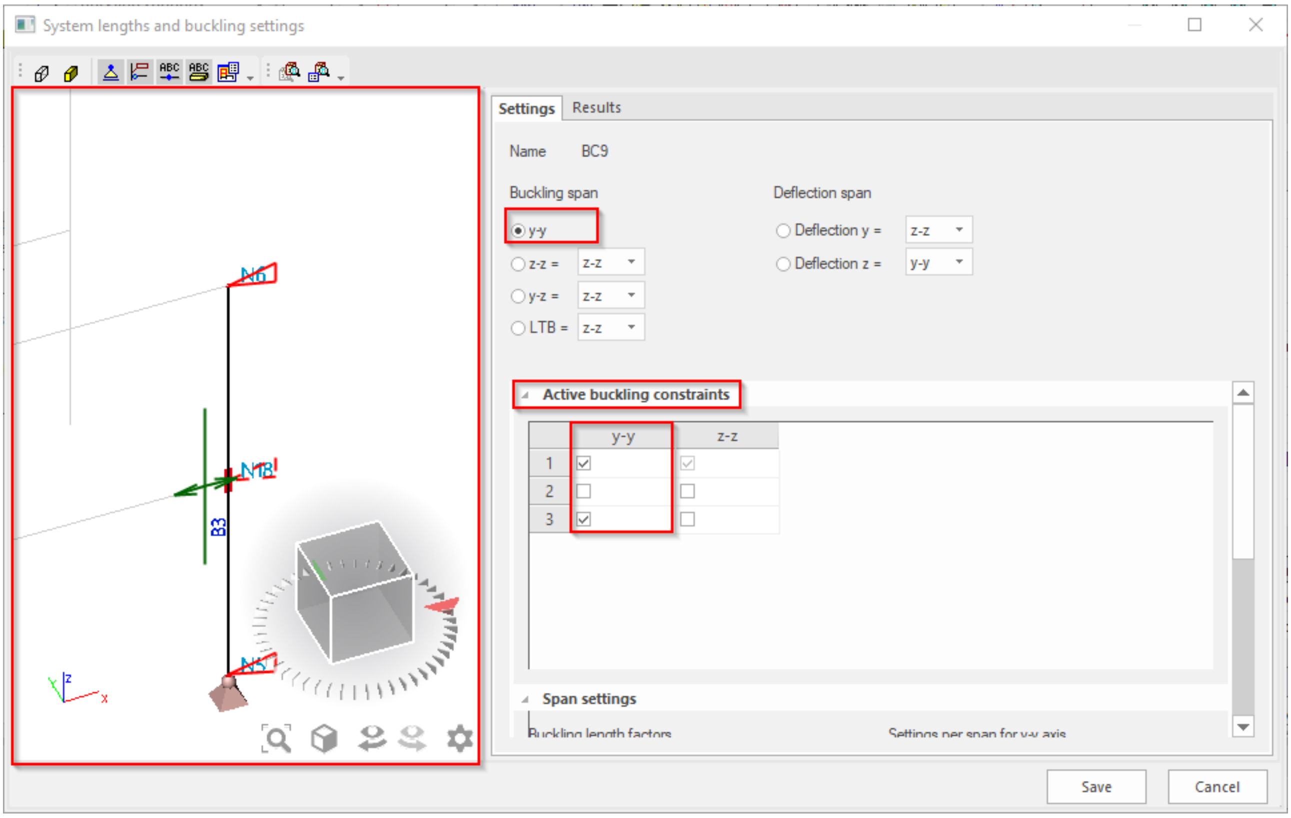
If the additional data (secondary member) is inserted, the buckling system of column B3 can be changed.
Select member B3 and change the System lengths and buckling settings back to 'Default'. Afterwards, beam B17 will not be included in the system lengths:





It is also possible to change the buckling supports using the option graphical window in the 'System lengths and buckling settings' menu an element. By clicking on the 3 dots, the System lengths and buckling settings window will open.

In the graphical window, the full red lines are the fixed nodes (i.e. active restraint), the dot lines are the free nodes (i.e. inactive restraint). By simply clicking on the triangles, the settings for free and fixed will be changed:



The user can do the same for the buckling span z-z. Also, in the previous window, it should be noticed that besides graphical input, buckling constraints can be ticked off/on using the ‘Active buckling constraints’.

These settings will be stored in the buckling system BC*. This buckling system can be used for other elements, but only of these elements that have the same amount of structural nodes.

Automatic calculation of buckling ratios

By default, the buckling ratios are calculated by SCIA Engineer. For this, two approximate formulas are used: one formula for a non sway structure (resulting in a buckling factor smaller (or equal) than 1) and one formula for a sway structure (resulting in a buckling factor higher (or equal) than 1):

- For a non sway structure:

$$k = \frac{(\rho_1\rho_2 + 5\rho_1 + 5\rho_2 + 24)(\rho_1\rho_2 + 4\rho_1 + 4\rho_2 + 12)2}{(2\rho_1\rho_2 + 11\rho_1 + 5\rho_2 + 24)(2\rho_1\rho_2 + 5\rho_1 + 11\rho_2 + 24)}$$

- For a sway structure:

$$k = x\sqrt{\frac{\pi^2}{\rho_1 x} + 4}$$

with	k	the buckling factor
	L	the system length
	E	the modulus of Young
	I	the moment of inertia
	C_i	the stiffness in node i
	M_i	the moment in node i
	ϕ_i	the rotation in node i

$$x = \frac{4\rho_1\rho_2 + \pi^2\rho_1}{\pi^2(\rho_1 + \rho_2) + 8\rho_1\rho_2}$$

$$\rho_i = \frac{C_i L}{EI}$$

$$C_i = \frac{M_i}{\phi_i}$$

The values for M_i and ϕ_i are approximately determined by the internal forces and the deformations, calculated by load cases which generate deformation forms, having an affinity with the buckling form.

The calculation of the k ratios is automatically done when calculating the structure linearly. For this, two additional load cases are calculated in the background:

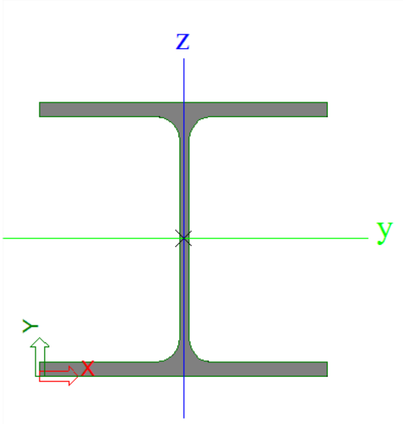
- Load case 1:
  - on the beams, the local distributed loads qy=1 N/m and qz=-100 N/m are used
  - on the columns the global distributed loads Qx =10000 N/m and Qy =10000 N/m are used.
- Load case 2:
  - on the beams, the local distributed loads qy=-1 N/m and qz=-100 N/m are used
  - on the columns the global distributed loads Qx =-10000 N/m and Qy=-10000 N/m are used.

Since these load cases, and thus the buckling ratios, are calculated during the linear calculation, it is necessary to always perform a linear calculation of the structure. **So when calculating non linear, the user should also perform a linear calculation otherwise no buckling factors are calculated and no steel code check can be performed.**

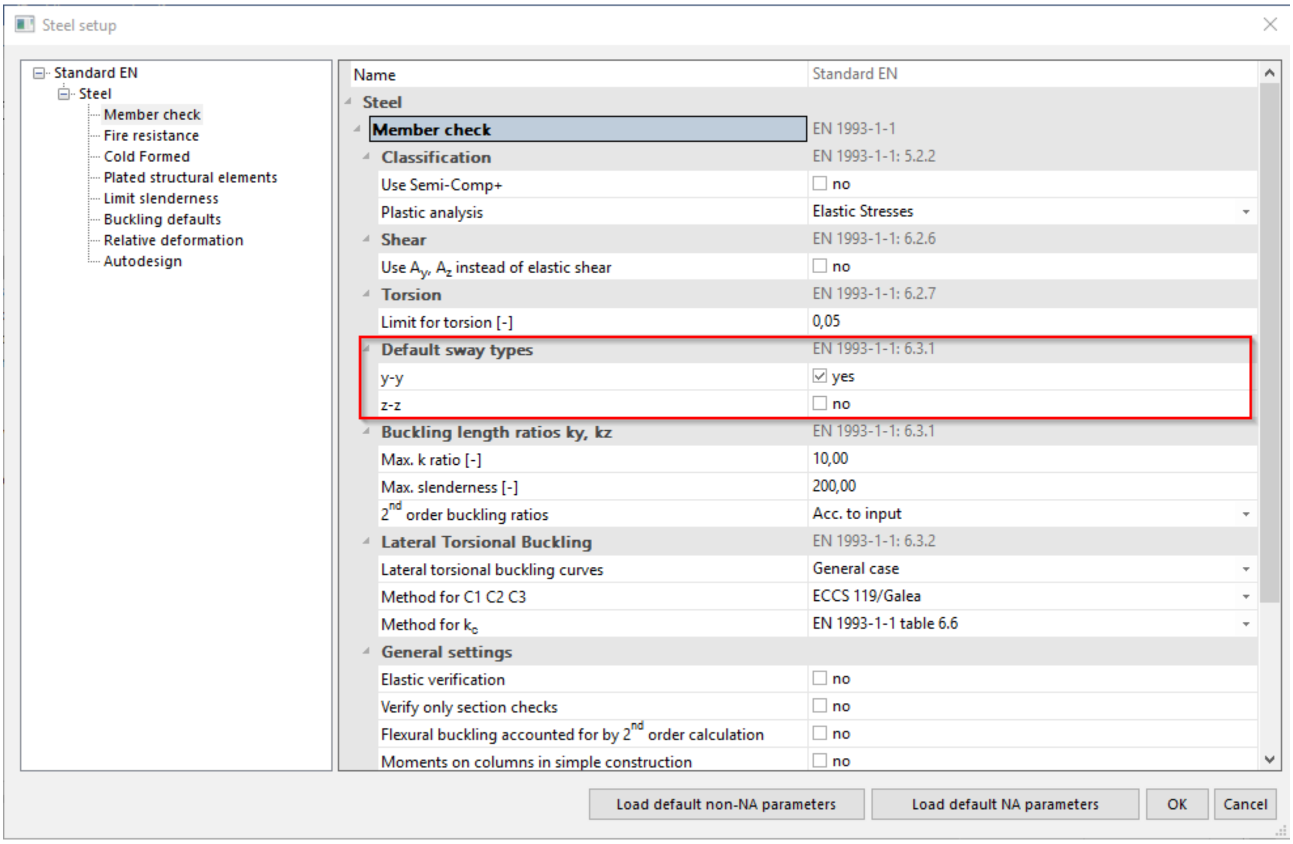
Note: The used approach gives good results for frame structures with perpendicular rigid or semi-rigid beam connections. For other cases, the user has to evaluate the presented bucking ratios.

Since the formulas are different for sway and non sway structure, it is important that the correct options are chosen for the two local directions:

- y-y: buckling around the local y-axis (so deformation in the direction of the local z-axis)
- z-z: buckling around the local z-axis (so deformation in the direction of the local y-axis)



The option ‘Sway’ or ‘Non sway’ can be chosen in the menu ‘Steel > Beams > Steel Setup’. These settings apply to the whole structure:



Example: Buckling Factor.esa

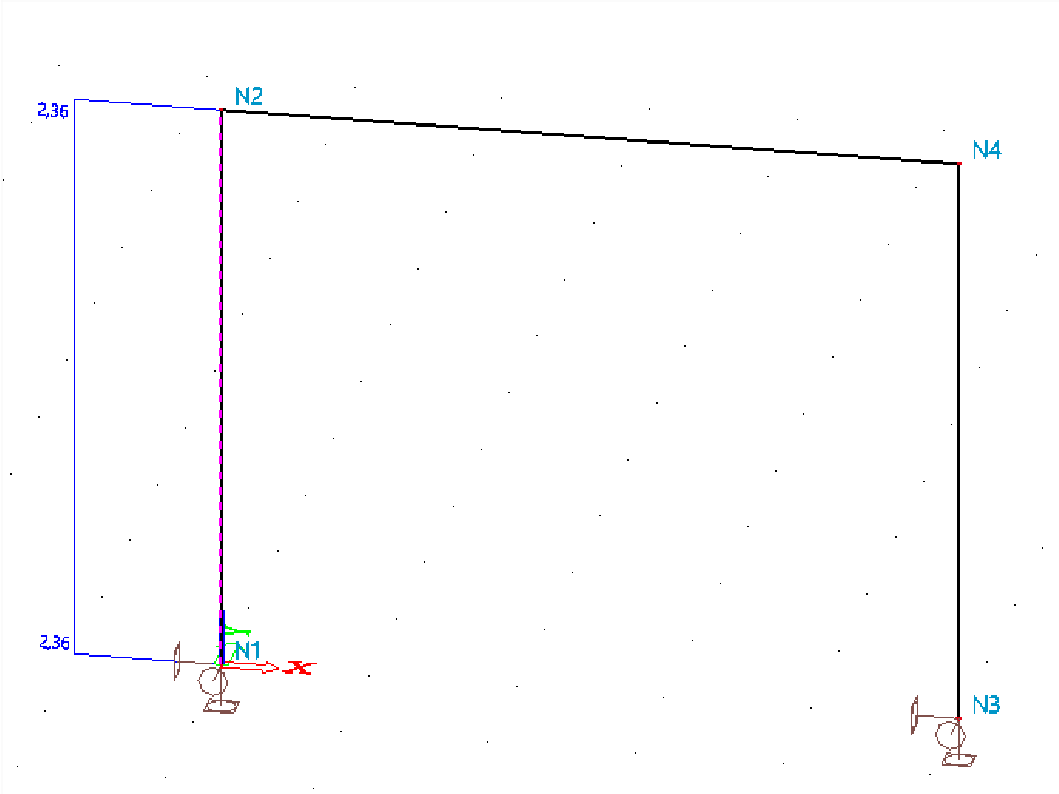
In this example, a simple steel frame is used to calculate the buckling ratios. The structure is loaded by the **two additional load cases** as described on the previous page.

Consider Column B1:

- L = 4000mm
- Set as sway
- In node N1 : My = 0 kNm => $C_2 = \rho_2 = 0.0$
- This node N1 defines ρ_2 because ρ_2 is always the smaller of the two.
- In node N2 for Load case LC1:
  - M_{y1} = 79,883 kNm
  - ϕ_1 = fiy = 1,523 mrad
  - $C_1 = M_{y1} / \phi_1$ = 79,883 kNm / 1,523 mrad = 52,451 kNm/mrad
    = 5,2451 x 10^{10} Nmm/rad
  - E = 210 000 N/mm²
  - Iy = 162700000 mm⁴
  - $\rho_1 = \frac{C_i L}{EI} = \frac{5.2451 \cdot \frac{10^{10} Nmm}{rad} \cdot 4000 mm}{210000 \frac{N}{mm^2} \cdot 162700000\ mm^4} = 6.141$
  - $x = \frac{4\rho_1\rho_2 + \pi^2\rho_1}{\pi^2(\rho_1+\rho_2)+8\rho_1\rho_2} = \frac{4 \cdot 6,141 \cdot 0,0 + \pi^2 \cdot 6,141}{\pi^2(6,141+0,0)+8 \cdot 6,141 \cdot 0,0} = 1.0$
  - $k = x\sqrt{\frac{\pi^2}{\rho_1 x} + 4} = 1,0\sqrt{\frac{\pi^2}{6,141 \cdot 1,0} + 4} = 2.368$
  - $N_{cr} = \frac{\pi^2 EI}{k^2 L^2} = \frac{\pi^2 \cdot 210000 N/mm^2 \cdot 162700000\ mm^4}{(2,368)^2 (4000)^2} = 3754765N = 3754,8\ kN$

This calculated buckling ratio can also be found in SCIA Engineer.

Under ‘Steel > Beams > Steel slenderness’, the buckling factor k_y will be found:



This can also be displayed in the preview:

Steel slenderness

Linear calculation

Member	CS Name	Part	Sway y	Ly [m]	ky [-]	ly [m]	Lam y [-]	lyz [m]	l LTB [m]
			Sway z	Lz [m]	kz [-]	lz [m]	Lam z [-]		
B1	CS1	1	Yes	4,000	2,36	9,459	63,23	4,000	4,000
			No	4,000	0,70	2,800	73,92		

With the option **Steel slenderness**, all parameters can be shown graphically and numerically in the preview.

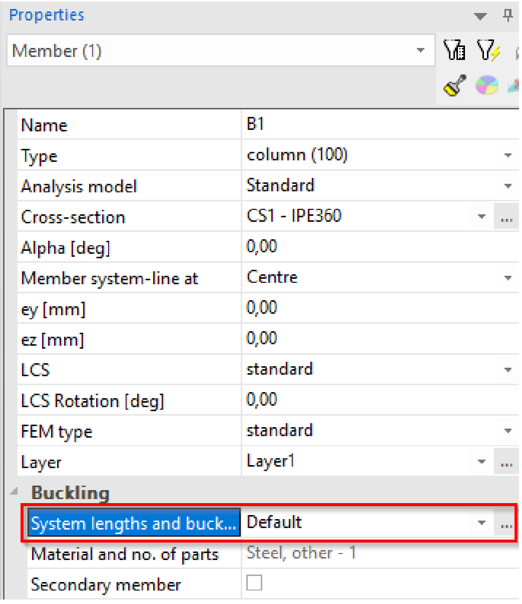
- Sway y (z): whether this member is considered sway or not sway.
- L_y (L_z): the system length of the member. This is the distance between the buckling supports.
- k_y (k_z): the buckling ratio. This factor is calculated automatically by SCIA Engineer or can be inserted manually.
- l_y (l_z): the buckling length.
  - $l_y = L_y * k_y$
- Lam y (Lam z): the slenderness of the element
- l_{yz} : the torsional buckling length
- l LTB: the lateral torsional buckling length

The type ‘sway or not sway’ in the steel setup applies for the whole structure. It is also possible to change these (overrule) these settings for each member (or selection of members) separately.

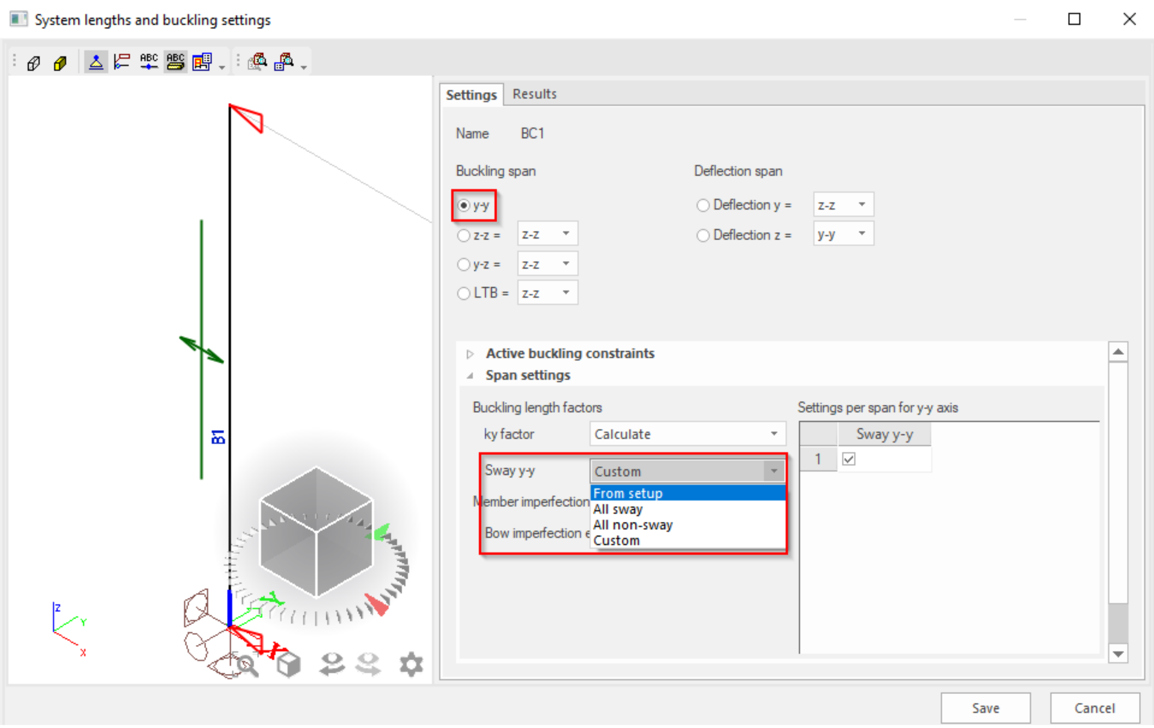
This can be done by using System lengths and buckling settings. Column B2 is used.

- **System lengths and buckling settings**

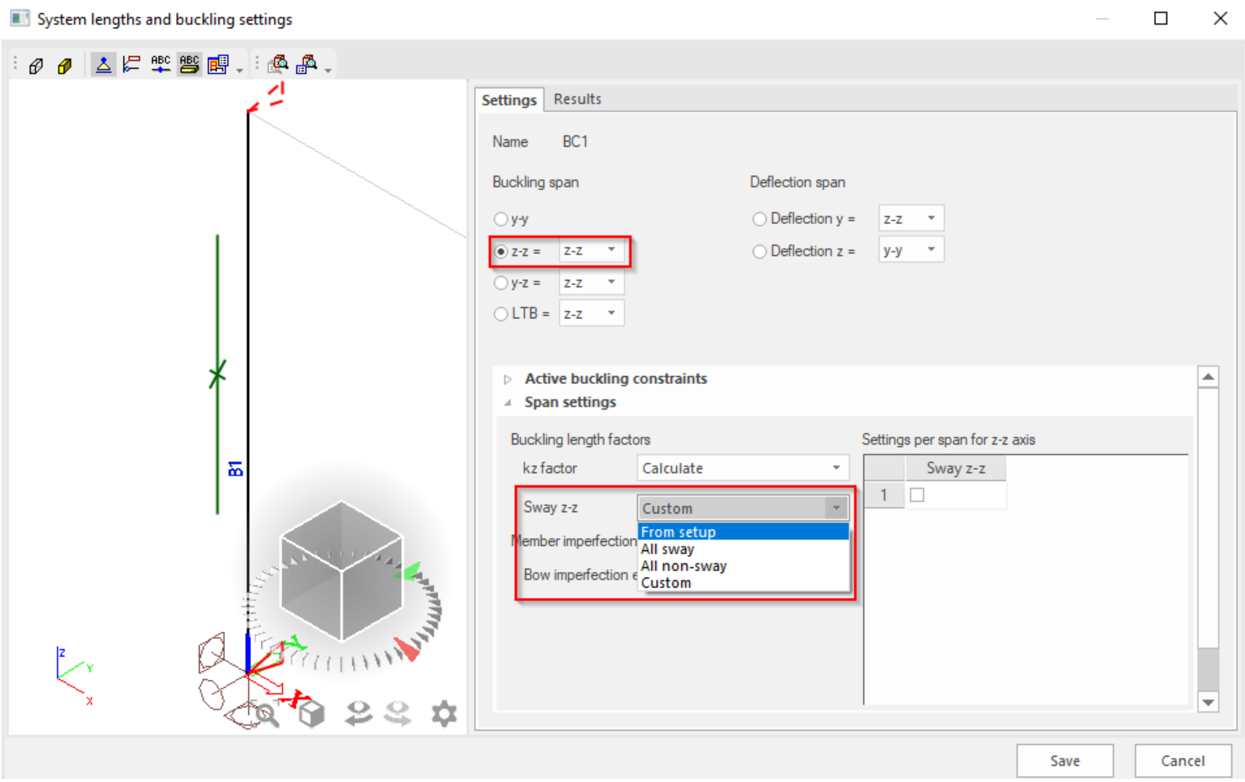
  ‘Property window of the selected member’.



By clicking on the three dots, the ‘System lengths and buckling settings’ window will open. Here the sway type can be changed.



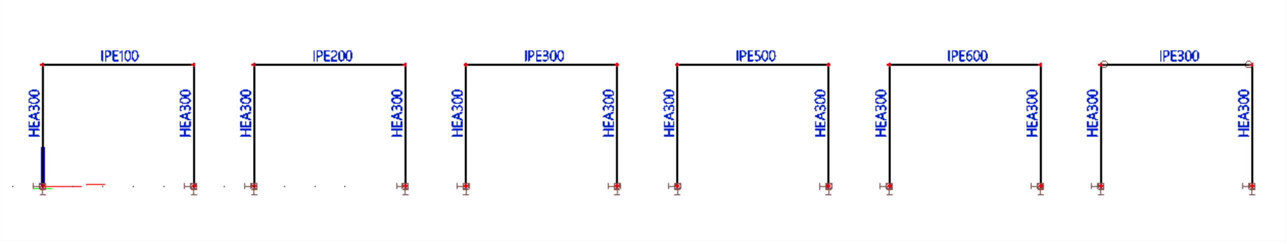
The same can be done for Sway z-z by changing Buckling span to z-z.



These settings are stored in the 'Buckling system' BC1. This system can be used for other members, if these members have the same amount of construction nodes.

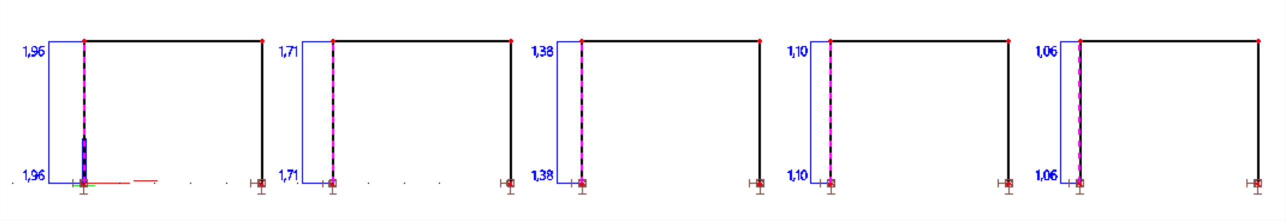
Example: Several frames.esa

In this example, several steel frames are modeled.
In each frame, the same type of column is used, but the beams are adapted. The connection between the columns is fixed. Only in the last frame, this connection is made hinged.

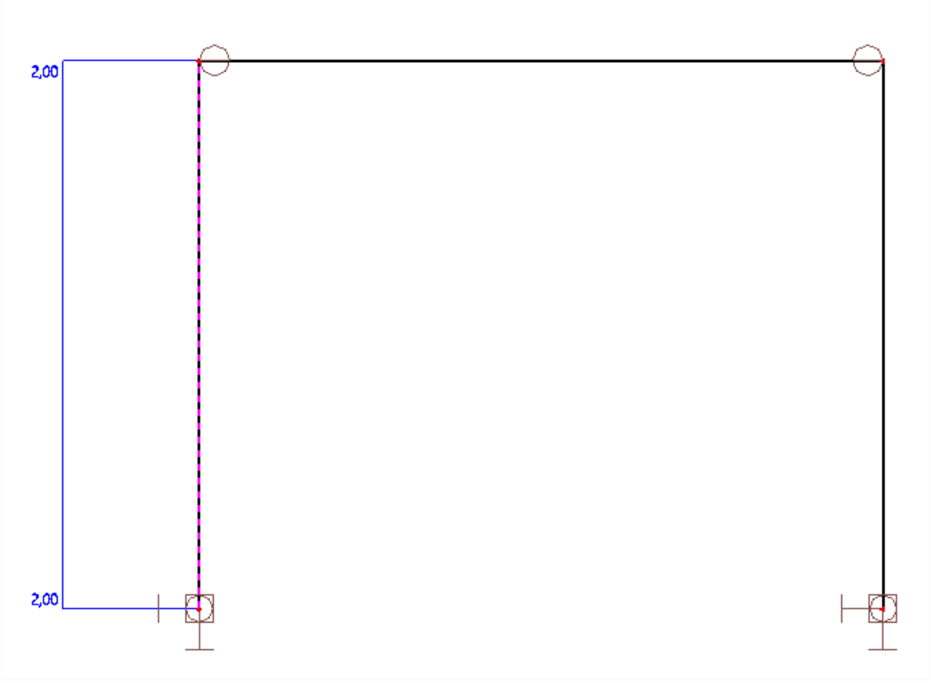


Since the structure is considered as 'sway' for y-y, it is expected to have a k_y factor between 1 and 2 for the columns.

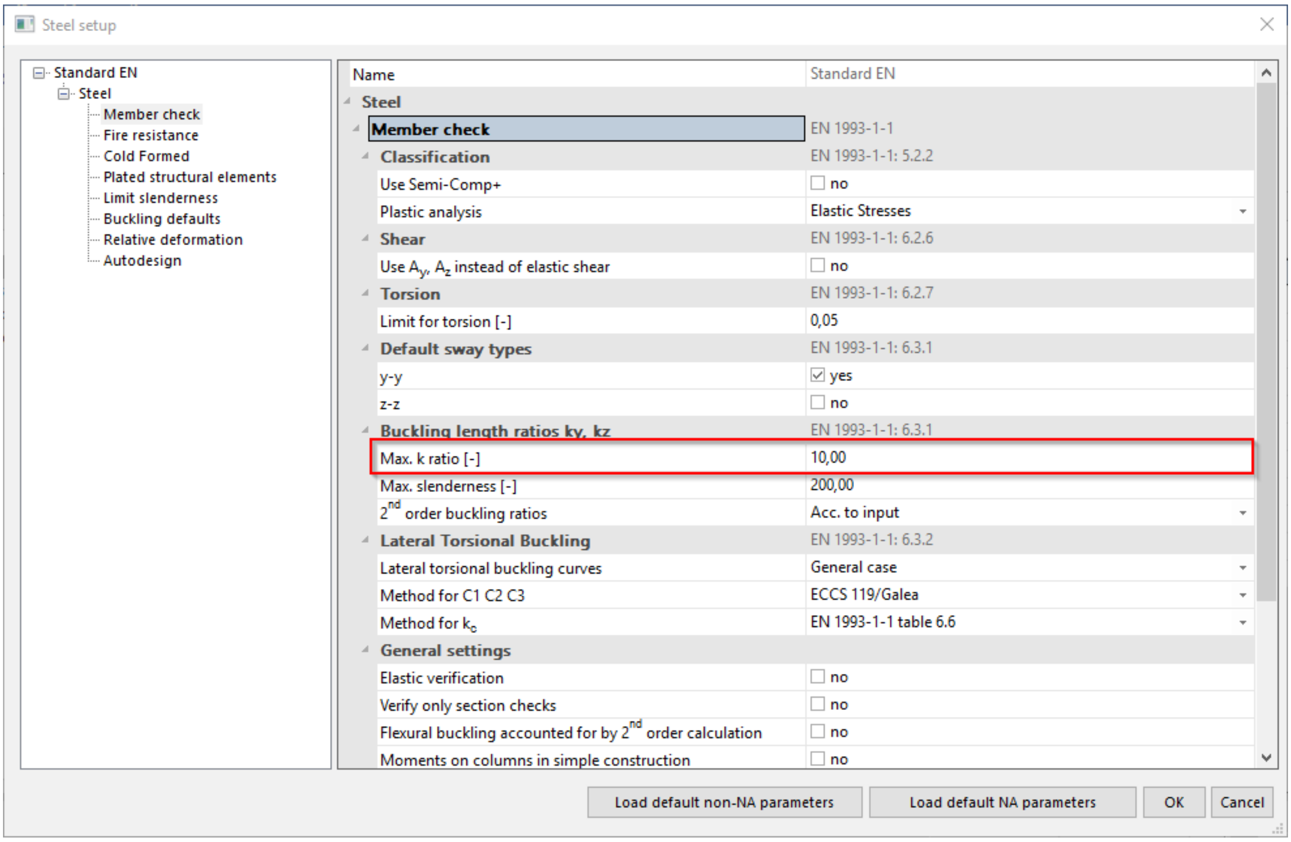
With a smaller beam, the k_y factor will be almost 2. With a bigger beam, the k_y factor will be almost 1.



In the frame with the hinged connection between the beams and the columns, the k_y factor of the columns will be equal to 2.



Note: If the buckling ratios are calculated automatically, by default a maximum value of 10 is used. This limitation can be adapted in the Steel setup.



Manual input of buckling ratios

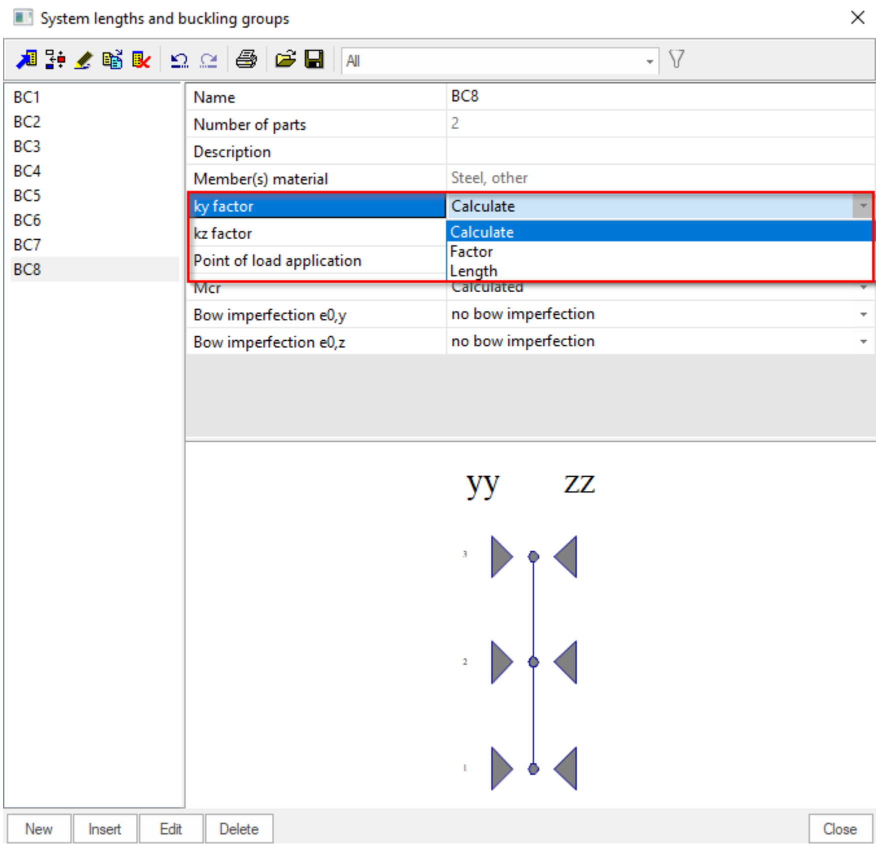
As mentioned in a previous chapter, the buckling ratios are automatically calculated by SCIA Engineer during the linear calculation. It is also possible to insert these buckling ratios manually.

This can be done using:

- System lengths and buckling groups.
  'Steel > Beams > System lengths and buckling groups.

  Column B4 will be used to explain this option.

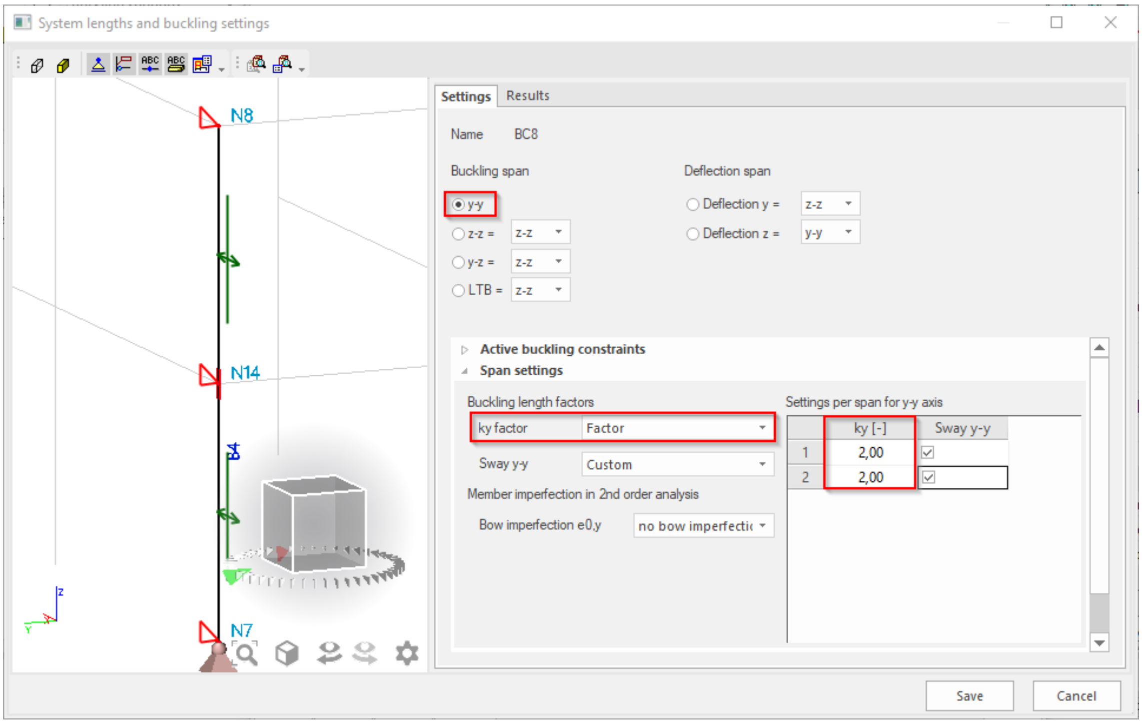
  Select member B4 and choose System lengths and buckling groups.



For the ky factor and the kz factor, there are three possibilities:

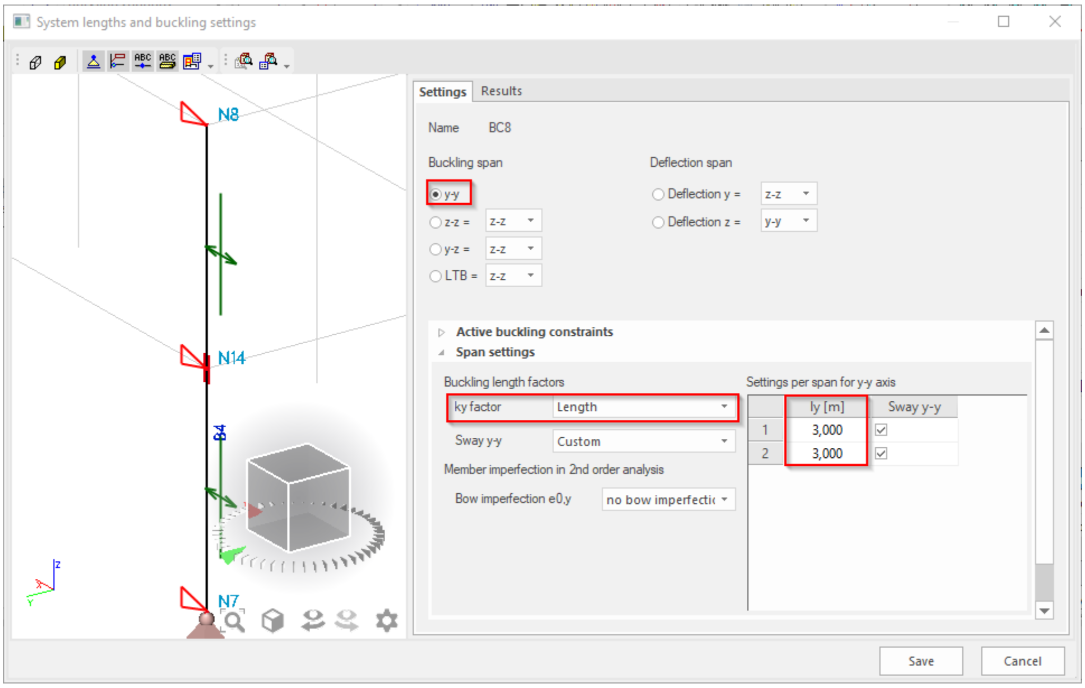
- Calculate: use the buckling ratio calculated by SCIA Engineer
- Factor: insert manually the buckling factor ky.
- Length: insert manually the buckling length ly.

To be able to insert these factors manually, the option **Edit** should be used and window System lengths and buckling settings will be opened for this column. In this window, the buckling ratio ky or the buckling length ly can be inputted.

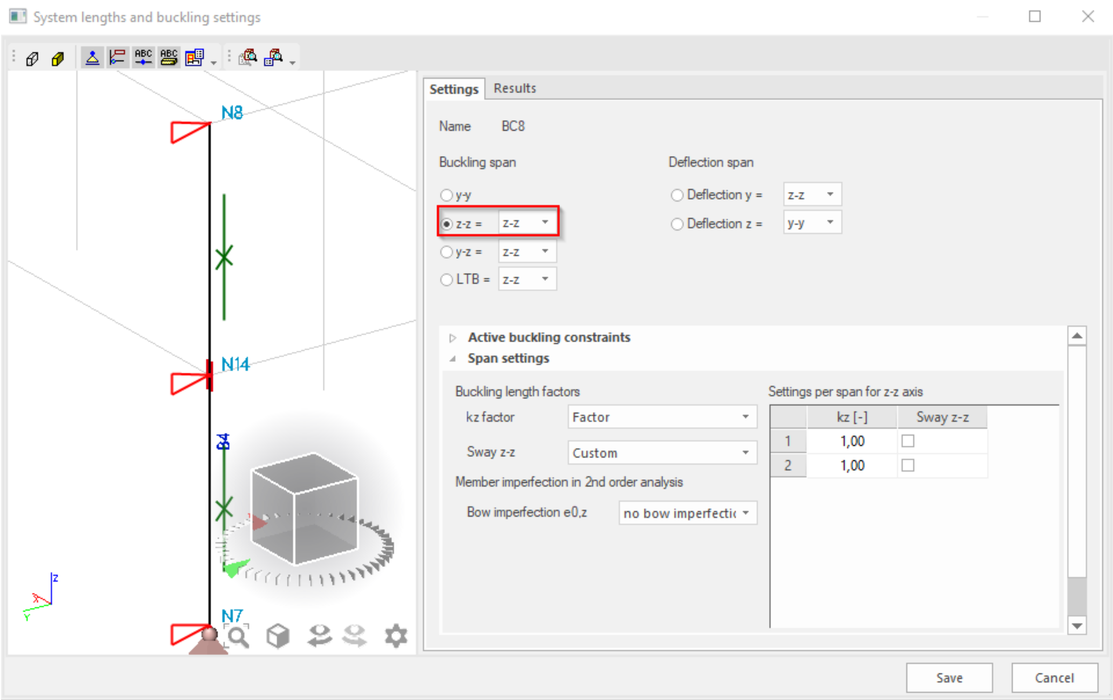


For this column B4, a ky ratio of 3 will be inserted manually. After inputting this value, a new label will appear on this column. Besides this, it is possible to insert a different buckling ratio for each part of the column, in this example both parts of the element, (1) and (2), have the same ky factor.

The user can insert the buckling length ly manually as well.



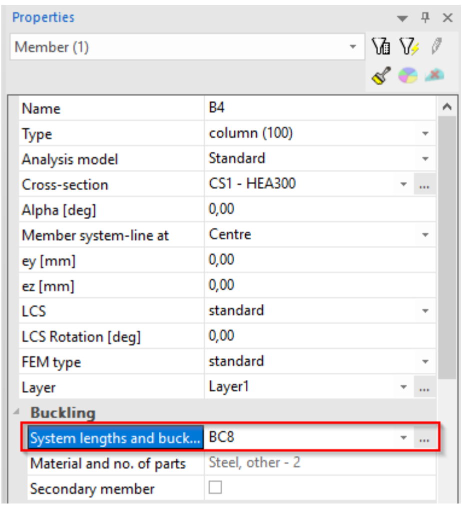
If the same wants to be done for kz and lz, the buckling span should be switched to z-z.



This buckling data is again additional data which can be copied to other elements.

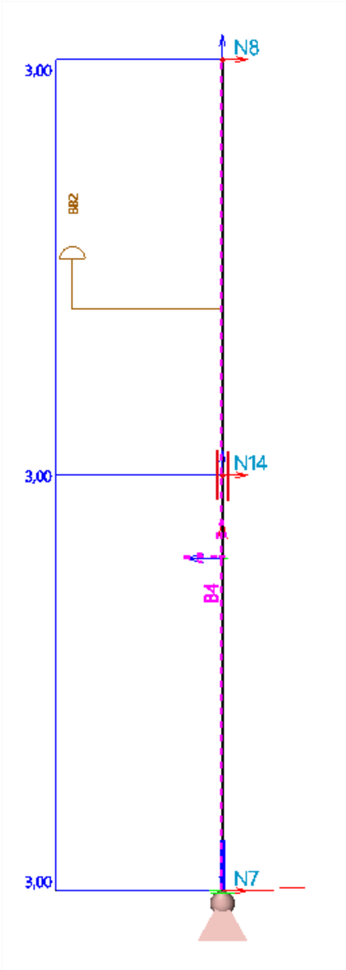
- System lengths and buckling settings.
  ‘Property window of the element’ B4.

  The same window explained in previous example can be directly accessed via Property window of the element B4.



You can check ky factors for column B4 in the steel menu: ‘Steel > Beams > Steel slenderness’

For column B4, the ky ratio of 3 is used:



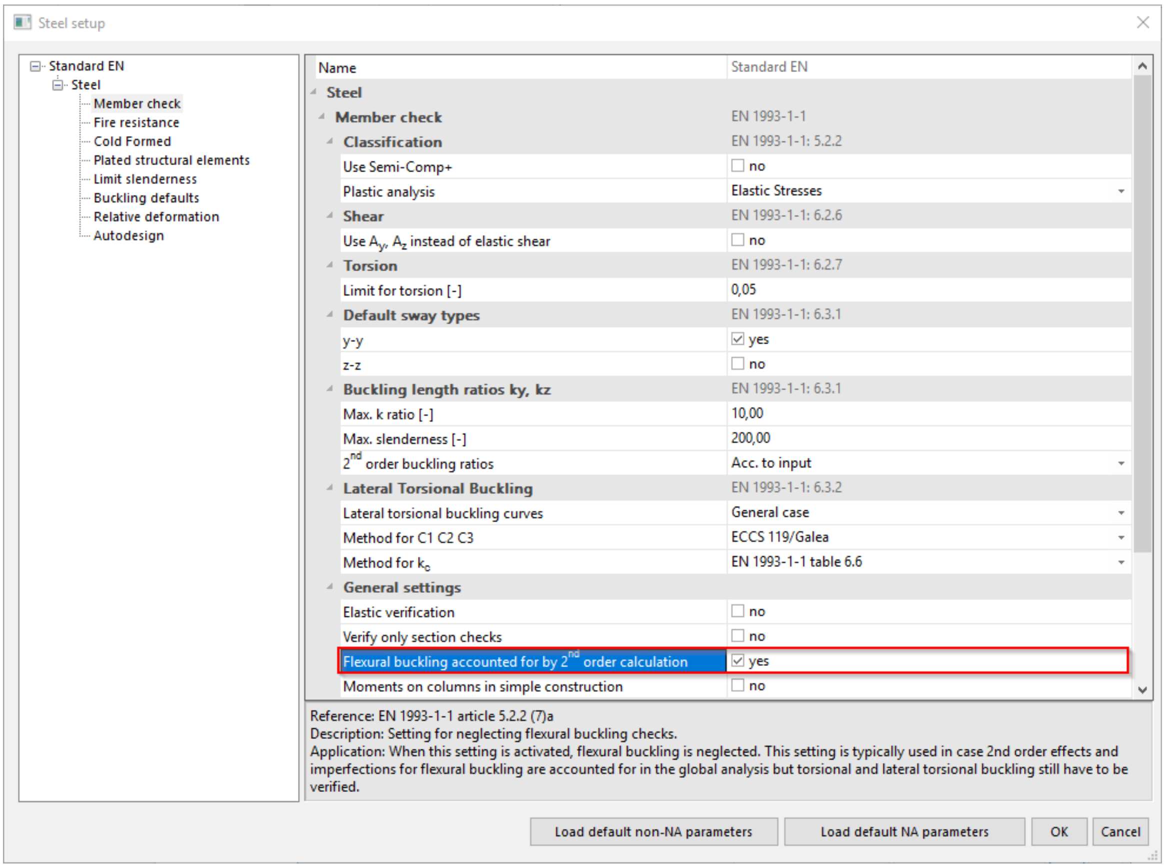
N8
3,00
BB2
N14
3,00
B4
N7
3,00

Buckling ratios using second order analysis

As mentioned in the previous chapters, the buckling ratios are automatically calculated by SCIA Engineer during the linear calculation. The second option for the user is to manually input the buckling ratios.

The third option is performing a second order analysis.
According to EN1993-1-1 chapter 5.2.2 (7) a), no individual stability check for the members according to 6.3 (= Buckling resistance of members) is necessary, if second order effects in individual members and relevant member imperfections are totally accounted for in the global analysis of the structure.

This particular article of the Eurocode is also implemented in SCIA Engineer. The user can activate this option in the general **Steel setup** in the Steel service:



By activating this option, SCIA Engineer will take the buckling ratios ky and kz equal to 0,001. This neglectable buckling ratios will influence the following two stability checks which are described in EN1993-1-1:

- **Flexural buckling check**

The use of a buckling ratio equal to 0,001 will ensure the fact that one or both conditions mentioned in EN1993-1-1 article 6.3.1.2 (4) are met:

(4) For slenderness $\bar{\lambda} \leq 0{,}2$ or for $\frac{N_{Ed}}{N_{cr}} \leq 0{,}04$ the buckling effects may be ignored and only cross sectional checks apply.

This remark will also be printed below the results table of the Flexural buckling check in the detailed output:

Flexural Buckling check
According to EN 1993-1-1 article 6.3.1.1 and formula (6.46)

Buckling parameters	yy	zz	
Sway type	sway	non-sway	
System length L	5,000	5,000	m
Buckling factor k	4,36	0,84	
Buckling length L_{cr}	21,813	4,201	m
Critical Euler load N_{cr}	160,74	1573,58	kN
Slenderness λ	263,38	84,18	
Relative slenderness λ_{rel}	2,80	0,90	
Limit slenderness $\lambda_{rel,0}$	0,20	0,20	

Note: The slenderness or compression force is such that Flexural Buckling effects may be ignored according to EN 1993-1-1 article 6.3.1.2(4).

- **Combined bending and axial compression check**

This check is executed according to formulas 6.61 and 6.62 of EN1993-1-1 article 6.3.3. Within this course, this check will not be discussed in detail since this a part of the Advanced Steel code check training, but the influence of the buckling ratios for this check will be explained.

$$\frac{N_{Ed}}{\frac{\chi_y N_{Rk}}{\gamma_{M1}}} + k_{yy}\frac{M_{y,Ed} + \Delta M_{y,Ed}}{\chi_{LT}\frac{M_{y,Rk}}{\gamma_{M1}}} + k_{yz}\frac{M_{z,Ed} + \Delta M_{z,Ed}}{\frac{M_{z,Rk}}{\gamma_{M1}}} \leq 1$$

$$\frac{N_{Ed}}{\frac{\chi_z N_{Rk}}{\gamma_{M1}}} + k_{zy}\frac{M_{y,Ed} + \Delta M_{y,Ed}}{\chi_{LT}\frac{M_{y,Rk}}{\gamma_{M1}}} + k_{zz}\frac{M_{z,Ed} + \Delta M_{z,Ed}}{\frac{M_{z,Rk}}{\gamma_{M1}}} \leq 1$$

If the buckling ratios are taken equal to 0.001, this will have a positive influence on the combined bending and axial compression check.

- X_y and X_z become equal to 1 => no reduction on the compression force check
- k_{yy}, k_{yz}, k_{zy} and k_{zz} will be reduced => positive influence on the bending check

Buckling ratios using stability analysis

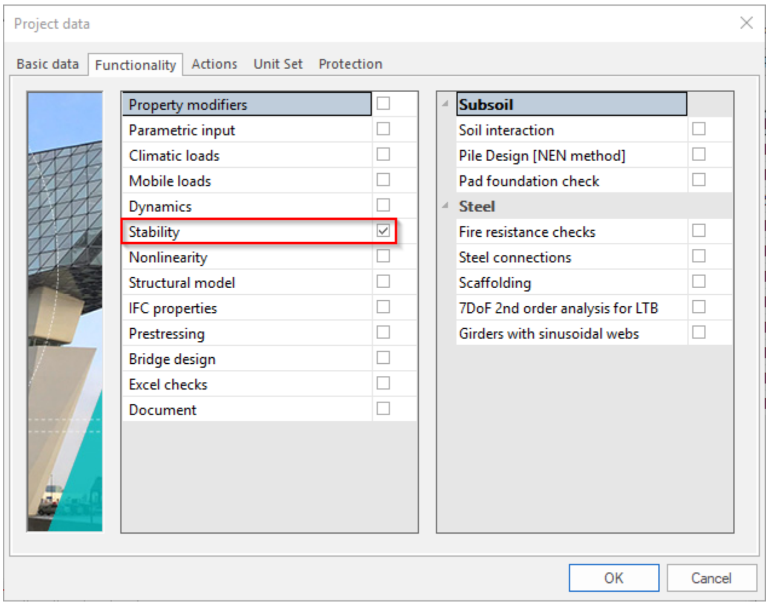
As mentioned in the previous chapters, the buckling ratios are automatically calculated by SCIA Engineer during the linear calculation. The second option for the user is to manually input the buckling ratios and the third option is to ignore the flexural buckling check when performing a second order analysis.

As a final option, the results of a stability analysis (buckling shape analysis) can be used to determine the buckling ratios. This type off analysis will be explained step by step in this chapter.

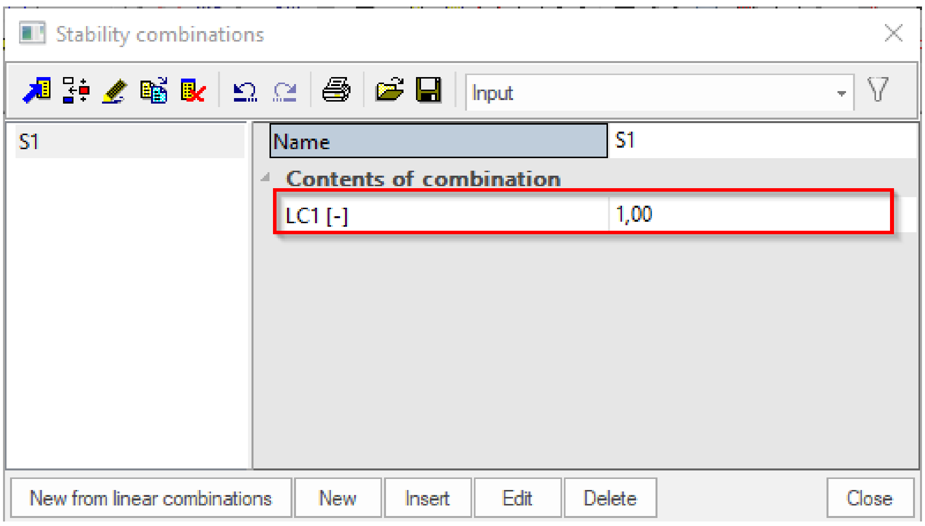
Example: Buckling Shape.esa

Open the example Buckling Shape. In this example, an IPE300 column is loaded with a unit compression force of 1kN. The following steps will explain how the stability analysis can be used to determine the buckling ratios in SCIA Engineer.

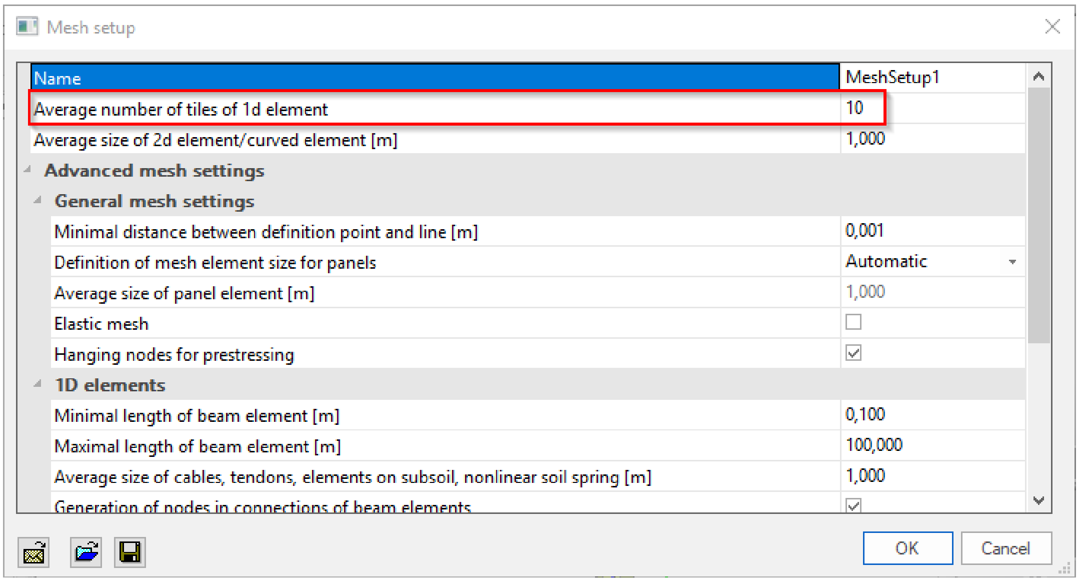
1) Activate the Functionality **Stability** within the Project data window:



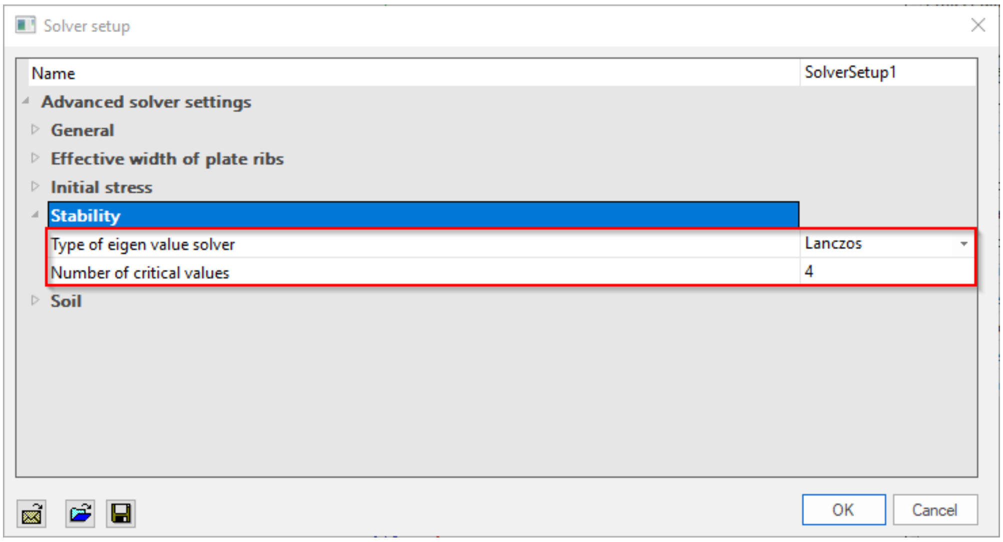
2) Create a Stability combination. This can be done in the service Load cases, Combinations => Stability combinations. In this example, the combination factor is kept equal to 1 with the consequence that the calculated critical load factor α_{cr} will be equal to the critical Euler force N_{cr}. This will be explained in one of the following steps.



3) Increase the number of mesh nodes along the column to be able to calculate the buckling shape in an accurate way (a minimum of 4 is recommended). In this example value is taken equal to 10.

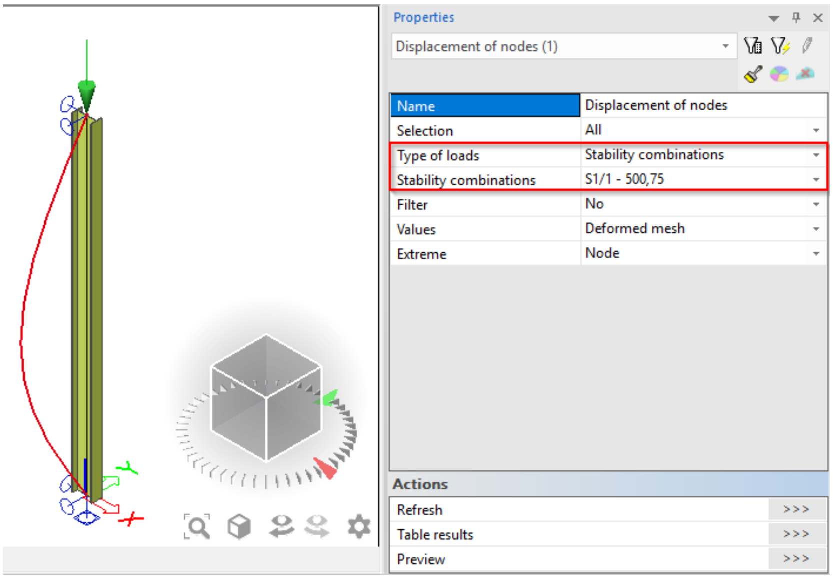


4) Determine the number of critical values to be calculated. In other words, determine how many theoretical buckling shapes have to be calculated. Since the buckling shapes are dependent on the stiffness of the elements in both local y and z direction, in this example we asked for 4 critical values. Also the type of eigen value solver can be chosen. More info about the different options can be found in the SCIA Engineer online help page: help.scia.net



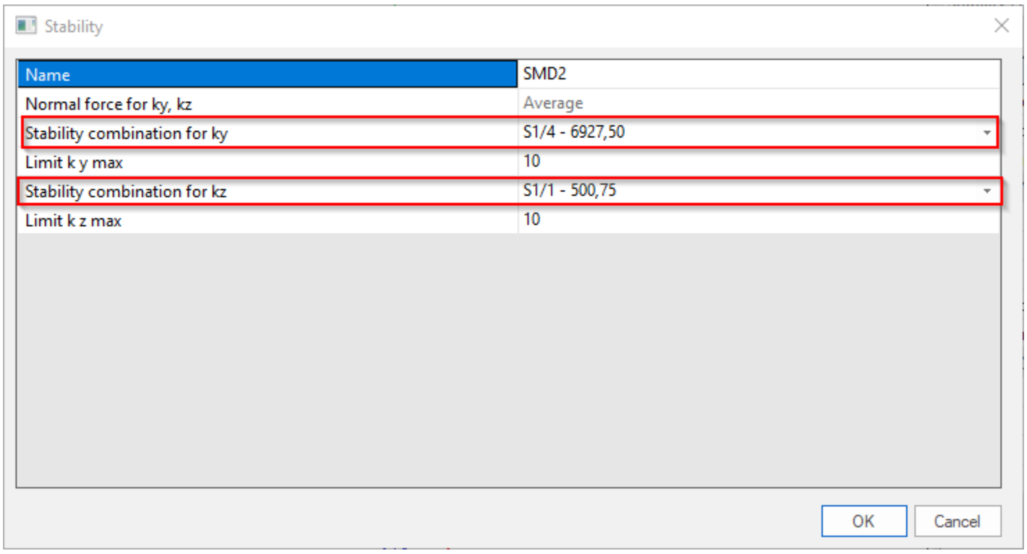
5) Run the Stability analysis

6) When the analysis is finished, the buckling shapes can be shown in the results service using "Displacements of nodes" asking for the "Deformed mesh" for the 4 calculated critical values.



The result clearly shows that the first buckling mode is a buckling mode around the weak z-axis (as expected). As a consequence, this buckling mode can be used for the calculation of the buckling ratio k_z. The same can be done for the buckling ratio k_y. In this particular example, the forth buckling mode is the decisive one for the strong y-axis buckling.

7) In the Steel service, the calculated buckling shapes can be inserted as data for the calculation of the buckling ratio's. This can be done under "Steel => Member Check data =>Stability member data". It is important to emphasize that the correct stability combinations are selected for ky and kz (i.e. these should induce major axis and minor axis buckling on this member).



Once defined, a label SMD1 will appear on this member.

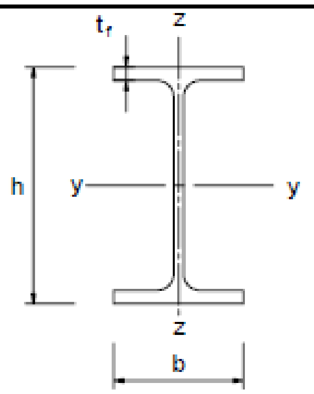
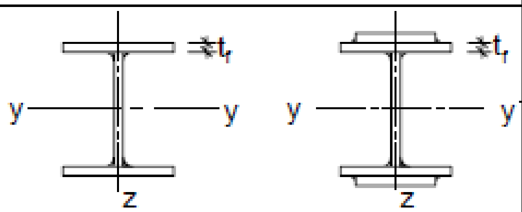
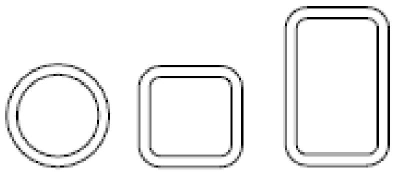
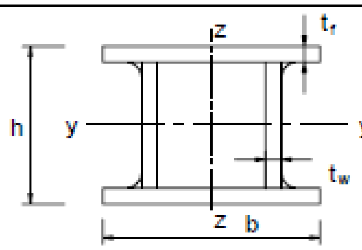
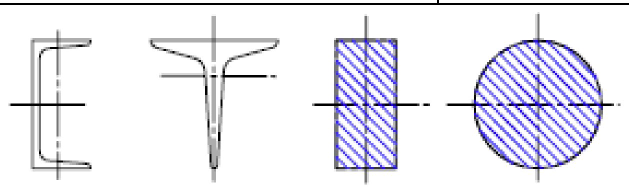
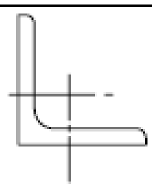
Remark: As mentioned in Step 2) the stability analysis calculates the critical load factor α_{cr}. This value can be used to determine if a structure can be designed with a first order analysis or need to be designed using a second order analysis taking into account global and or local imperfections as described in EN1993-1-1 article 5.2.1(3):

$$\alpha_{cr} = \frac{F_{cr}}{F_{Ed}} \geq 10 \quad \text{for elastic analysis}$$

Buckling curves

SCIA Engineer will use the buckling curves according to the code. If the Eurocode is chosen, table 6.2 of EN 1993-1-1 [1] will be used to determine which buckling curve has to be used.

Table 6.2: Selection of buckling curve for a cross-section

Cross section		Limits		Buckling about axis	Buckling curve S 235 S 275 S 355 S 420	Buckling curve S 460
Rolled sections		$h/b > 1{,}2$	$t_f \leq 40$ mm	y – y z – z	a b	a_0 a_0
			40 mm < $t_f \leq 100$	y – y z – z	b c	a a
		$h/b \leq 1{,}2$	$t_f \leq 100$ mm	y – y z – z	b c	a a
			$t_f > 100$ mm	y – y z – z	d d	c c
Welded I-sections			$t_f \leq 40$ mm	y – y z – z	b c	b c
			$t_f > 40$ mm	y – y z – z	c d	c d
Hollow sections			hot finished	any	a	a_0
			cold formed	any	c	c
Welded box sections			generally (except as below)	any	b	b
			thick welds: $a > 0{,}5t_f$ $b/t_f < 30$ $h/t_w < 30$	any	c	c
U-, T- and solid sections				any	c	c
L-sections				any	b	b

The automatic use of the correct buckling curve according to the code is supported for the following cross section types:

- Profile Library
- Haunch
- Sheet welded
- Build-in beams
- Thin-walled geometric
- Fabricated

For all other cross section types, the buckling curves for both directions are by default set to d. This can be changed manually by changing the properties of the cross section.

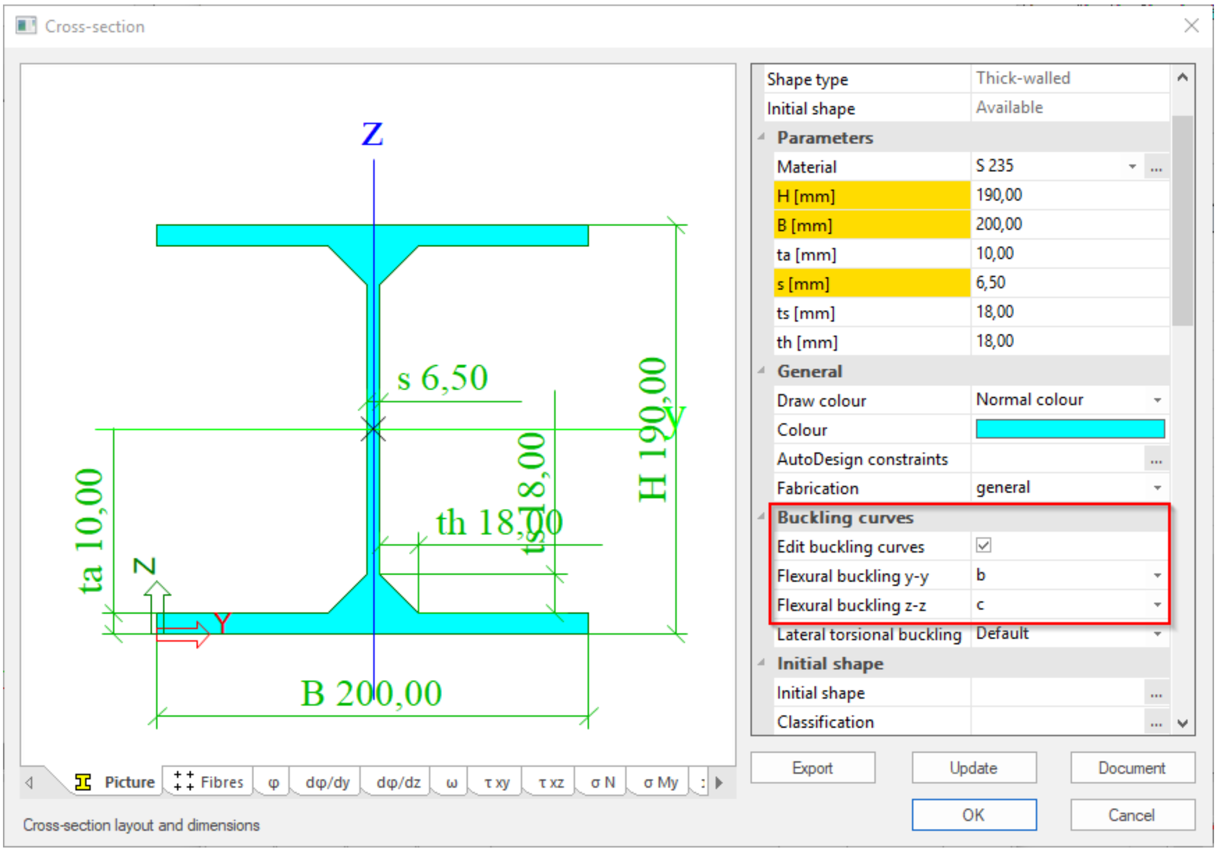
Example: Buckling curves.esa

Open the example Buckling curves. In this example, 3 columns are inserted with a fixed support and loaded with a point force of -100 kN.

The following cross sections are used:

- Profile Library > HEA200
- Geometric shape > default buckling curves d
- Geometric shape > buckling curves manually changed

By default, the buckling curves d are used for both directions for the second and third cross section. For the third cross section, this is manually changed into buckling curve b around the y axis and buckling curve c around the z axis. This way, the same curves are used as for the HEA200 profile.



Note: There have been some changes since SCIA Engineer version 2013.1.

- SCIA Engineer 2013.0 or older.

  In these versions, it is also possible to manually change the buckling curves for each cross section type. However, if a cross section from the profile library is used, the buckling check will always use the buckling curves according to table 6.2 of EN 1993-1-1 [1]. This means that for these types of cross section, the user input for the buckling curves is neglected in the Steel code check.

- SCIA Engineer 2013.1 or newer.

  The manual modification of the buckling curves in the cross section properties, is always taken into account by the buckling check in the steel menu, also for the cross sections from the profile library.

Buckling check

If the Eurocode is chosen, the flexural buckling check will be executed following EN 1993-1-1 art. 6.3.1. [1]

The compression member should be checked against buckling as follows:

$$\frac{N_{Ed}}{N_{b,Rd}} \leq 1$$

Where

- N_{Ed} is the design value of the compression force.
- $N_{b,Rd}$ is the design buckling resistance of the buckling member.

Note: In SCIA Engineer, a negative value of N_{Ed} means a compression force.

- $N_{b,Rd} = \frac{\chi \cdot A \cdot f_y}{\gamma_{M1}}$ For class 1, 2 or 3 cross sections
- $N_{b,Rd} = \frac{\chi \cdot A_{eff} \cdot f_y}{\gamma_{M1}}$ For class 4 cross sections

The reduction factor χ will be calculated as follows:

$$\chi = \frac{1}{\Phi + \sqrt{\Phi^2 - \bar{\lambda}^2}} \quad \text{but } \chi \leq 1{,}0$$

With

Φ $\quad 0{,}5\left[1 + \alpha\left(\bar{\lambda} - 0{,}2\right) + \bar{\lambda}^2\right]$

$\bar{\lambda}$ $\quad \sqrt{\frac{A \cdot f_y}{N_{cr}}}$ For class 1, 2 or 3 cross-sections

$\quad \sqrt{\frac{A_{eff} \cdot f_y}{N_{cr}}}$ For class 4 cross-sections

α Imperfection depending on the buckling curves:

Table 6.1: Imperfection factors for buckling curves

Buckling curve	a_0	a	b	c	d
Imperfection factor α	0,13	0,21	0,34	0,49	0,76

N_{cr} Critical normal force (Euler force)

$$N_{cr} = \frac{\pi^2 EI}{k^2 L^2}$$

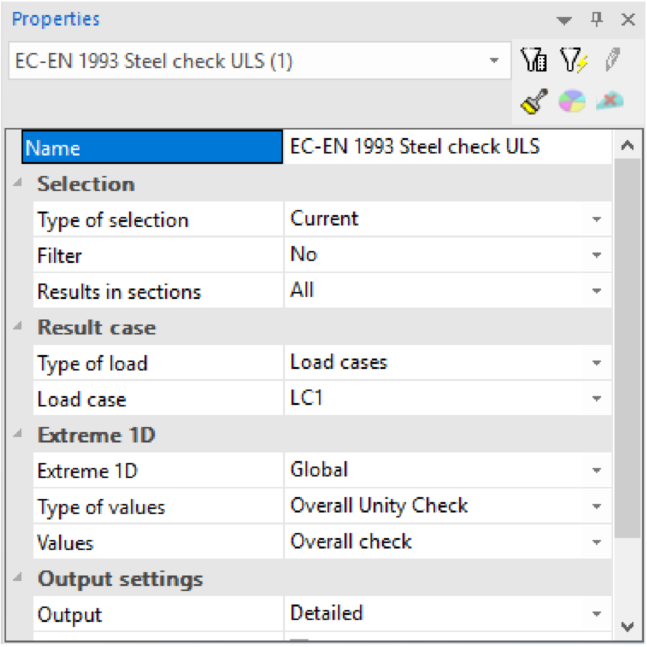
Note: For slenderness $\bar{\lambda} \leq 2$ or for $\frac{N_{Ed}}{N_{cr}} \leq 0{,}04$ the buckling effects may be ignored and only the cross sectional checks need to be applied.

Example: Buckling check.esa

Open the example Buckling check. In this example, a simple steel hall is modelled. The structure is loaded by the self weight and with a surface load of -1 kN/m² on the roof.

After executing the linear calculation, the buckling check can be performed in the Steel menu. ‘Steel > Beams > ULS Checks – EC – EN 1993 Steel Check ULS’.

Select column B4. In the property window of the ULS check, the following properties are chosen:



The selection is set to ‘Current’ to only see the results of the selected member (column B4).
The flexural buckling check will be performed in the **Stability check** and can be shown in the ‘Detailed’ output.

- For load LC1 (self weight):

Flexural Buckling check
According to EN 1993-1-1 article 6.3.1.1 and formula (6.46)

Buckling parameters	yy	zz	
Sway type	sway	non-sway	
System length L	5,000	5,000	m
Buckling factor k	4,38	0,82	
Buckling length L_{cr}	21,923	4,081	m
Critical Euler load N_{cr}	159,12	1667,78	kN
Slenderness λ	264,72	81,77	
Relative slenderness λ_{rel}	2,82	0,87	
Limit slenderness $\lambda_{rel,0}$	0,20	0,20	

Note: The slenderness or compression force is such that Flexural Buckling effects may be ignored according to EN 1993-1-1 article 6.3.1.2(4).

For load case LC1, the buckling check does not need to be performed since:

$$\frac{N_{Ed}}{N_{cr}} = \frac{3{,}40}{160{,}24} = 0{,}021 \leq 0{,}04$$

- For load LC2 (surface load):

Flexural Buckling check
According to EN 1993-1-1 article 6.3.1.1 and formula (6.46)

Buckling parameters	yy	zz	
Sway type	sway	non-sway	
System length L	5,000	5,000	m
Buckling factor k	4,38	0,82	
Buckling length L_{cr}	21,923	4,081	m
Critical Euler load N_{cr}	159,12	1667,78	kN
Slenderness λ	264,72	81,77	
Relative slenderness λ_{rel}	2,82	0,87	
Limit slenderness $\lambda_{rel,0}$	0,20	0,20	
Buckling curve	b	c	
Imperfection α	0,34	0,49	
Reduction factor χ	0,11	0,62	
Buckling resistance $N_{b,Rd}$	141,30	781,28	kN

Warning: Slenderness 264,72 is larger than the limit value of 200,00.

Flexural Buckling verification		
Cross-section area A	5,3800e-03	m²
Buckling resistance $N_{b,Rd}$	141,30	kN
Unity check	0,10	-

For this load case, the buckling check needs to be performed.

All the parameters needed to calculate the buckling resistance are shown in this table.
The buckling resistance is calculated for both the y and z direction. The smallest of these two values is used to perform the check.

- The buckling length = system length * buckling factor

  $l_y = k_y * L_y = 4{,}38 * 5 = 21{,}9$ m

- This buckling length is used to calculate the critical normal force (Euler force)

$$N_{cr} = \frac{\pi^2 EI}{k^2 L^2} = \frac{\pi^2 EI_y}{{l_y}^2} = \frac{\pi^2 * 2{,}1 * 10^8 \frac{kN}{m^2} * 3{,}69 * 10^{-5}\, m^4}{21{,}9^2\, m^2} = 159{,}46\, kN$$

- Slenderness

$$\lambda = \frac{l_y}{i_y} = \frac{21{,}9m}{0{,}082817m} = 264{,}44$$

- Relative slenderness

$$\bar{\lambda} = \frac{\lambda}{\lambda_1} = \frac{\lambda}{93{,}3\varepsilon} = \frac{264{,}44}{93{,}3} = 2{,}83$$

- Limit slenderness

  The limit slenderness is by default taken from EN 1993-1-1, article 6.3.2.3

  For flexural buckling, this is a constant value of 0,2.

- The buckling curve b is used, and thus an imperfection factor α of 0,34.
- The reduction factor χ will be calculated as follows:

$$\chi = \frac{1}{\Phi + \sqrt{\Phi^2 - \bar{\lambda}^2}}$$

And

$$\Phi = 0{,}5\left[1 + \alpha(\bar{\lambda} - 0{,}2) + \bar{\lambda}^2\right] = 0{,}5\left[1 + 0{,}34\,(2{,}83 - 0{,}2) + 2{,}83^2\right] = 4{,}95155$$

Thus

$$\chi = \frac{1}{\Phi + \sqrt{\Phi^2 - \bar{\lambda}^2}} = \frac{1}{4{,}95155 + \sqrt{4{,}95155^2 - 2{,}83^2}} = 0{,}1109$$

- Buckling resistance

$$N_{b,Rd} = \frac{\chi \cdot A \cdot f_y}{\gamma_{M1}} = \frac{0{,}1109 * 0{,}00538\ m^2 * 235000\ kN/m^2}{1} = 140{,}2109\ kN$$

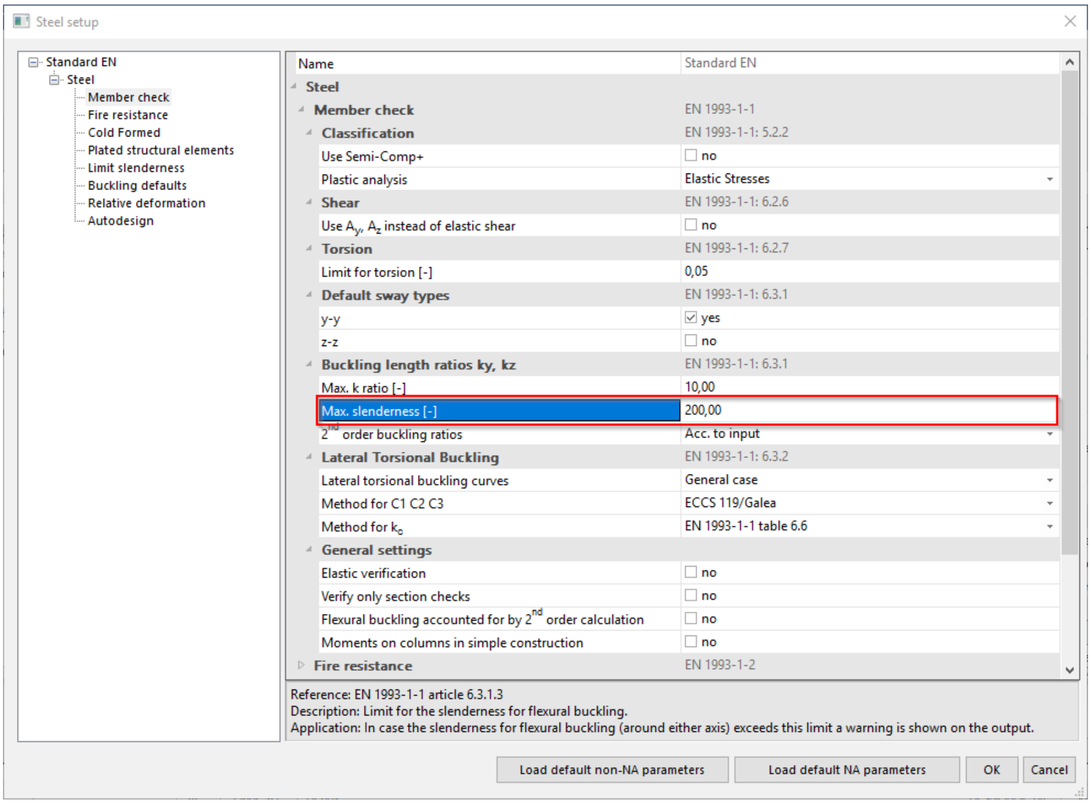
The difference with the result in SCIA Engineer is caused by rounding errors.

- The buckling check

$$\frac{N_{Ed}}{N_{b,Rd}} = \frac{13{,}53}{140{,}21} = 0{,}096$$

- There is also a warning message printed in the output of the Flexural Buckling Check about the slenderness of the member. In the steel setup, a maximal slenderness can be inserted. If the real slenderness of the member is higher than this inserted slenderness, this warning message will be printed.

In SCIA Engineer, by default the maximal slenderness is set to 200:



Relative deformation

Nodal displacements

First of all, a simple manual calculation can be done to check the nodal displacements in horizontal vertical direction.

The following limits can be checked:

- h/150 for horizontal displacements
- L/200 for vertical displacements

These values are guidelines.

Example: Buckling check.esa

Open the example Buckling check. Since this steel hall is only loaded with the self weight of the structure and a vertical surface load, the nodal displacements in the global Y direction are zero. So only the nodal displacements in the global X and Z directions will be checked.

To check these values, the load combination CO1 is used.

Displacement of nodes

Linear calculation, Extreme : Global
Selection : All
Combinations : CO1

Node	Case	Ux [mm]	Uy [mm]	Uz [mm]	Fix [mrad]	Fiy [mrad]	Fiz [mrad]
N6	CO1/1	**-36,9**	**0,0**	-0,2	0,1	8,2	3,2
N8	CO1/1	**37,0**	0,0	-0,2	0,3	-8,2	-3,2
N16	CO1/1	-36,9	**0,0**	-0,2	-0,1	8,2	-3,2
N7	CO1/1	0,1	0,0	**-223,7**	-0,9	0,0	0,0
N1	CO1/1	0,0	0,0	**0,0**	1,0	-7,1	6,1
N3	CO1/1	0,0	0,0	-103,9	**-65,7**	0,0	0,1
N22	CO1/1	0,0	0,0	-103,9	**65,7**	0,0	-0,1
N10	CO1/1	0,0	0,0	0,0	0,1	**-15,0**	8,2
N9	CO1/1	0,0	0,0	0,0	0,0	**15,0**	**-9,6**
N19	CO1/1	0,0	0,0	0,0	0,0	15,0	**9,6**

- Horizontal deformation

  The maximal nodal displacement in the global X direction is 37,0 mm.

  h/150 = 5000mm/150 = 33,33 mm

  37,0 mm > 33,33 mm so this is not ok.

- Vertical deformation

  The maximal nodal displacement in the global Z direction is 223,7 mm.

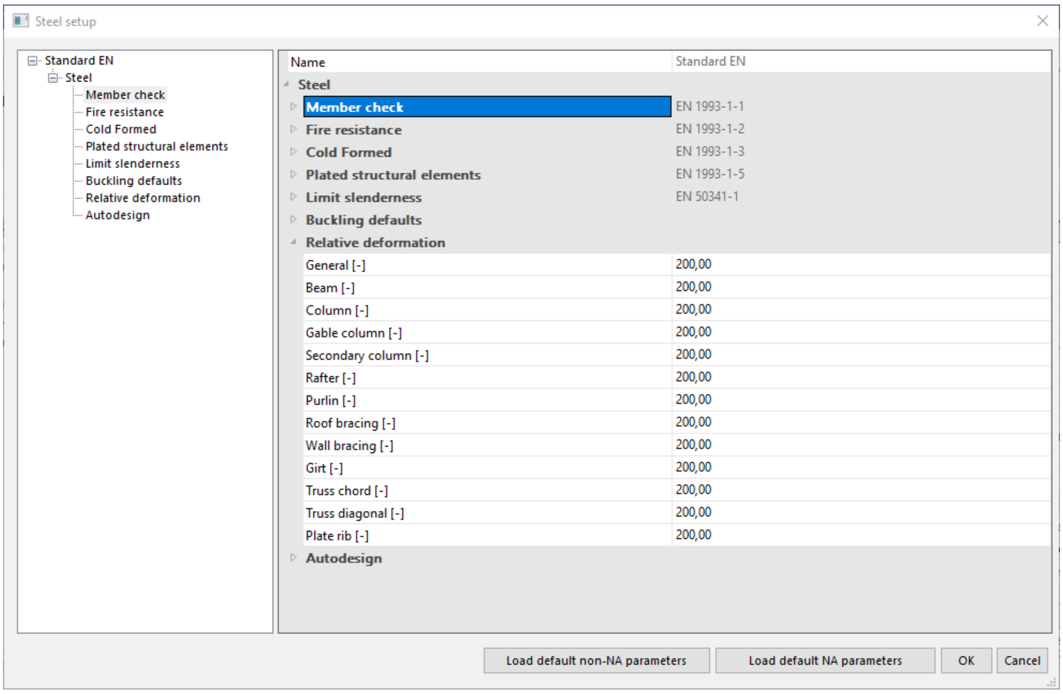
  L/200 = 12000/200 = 60 mm

  223,7 mm > 60 mm so this is not ok.

Relative deformation

For the relative deformation check, limiting values are inserted in the steel setup for each type of member.

‘Steel > Beams > Steel setup > Relative deformation’.

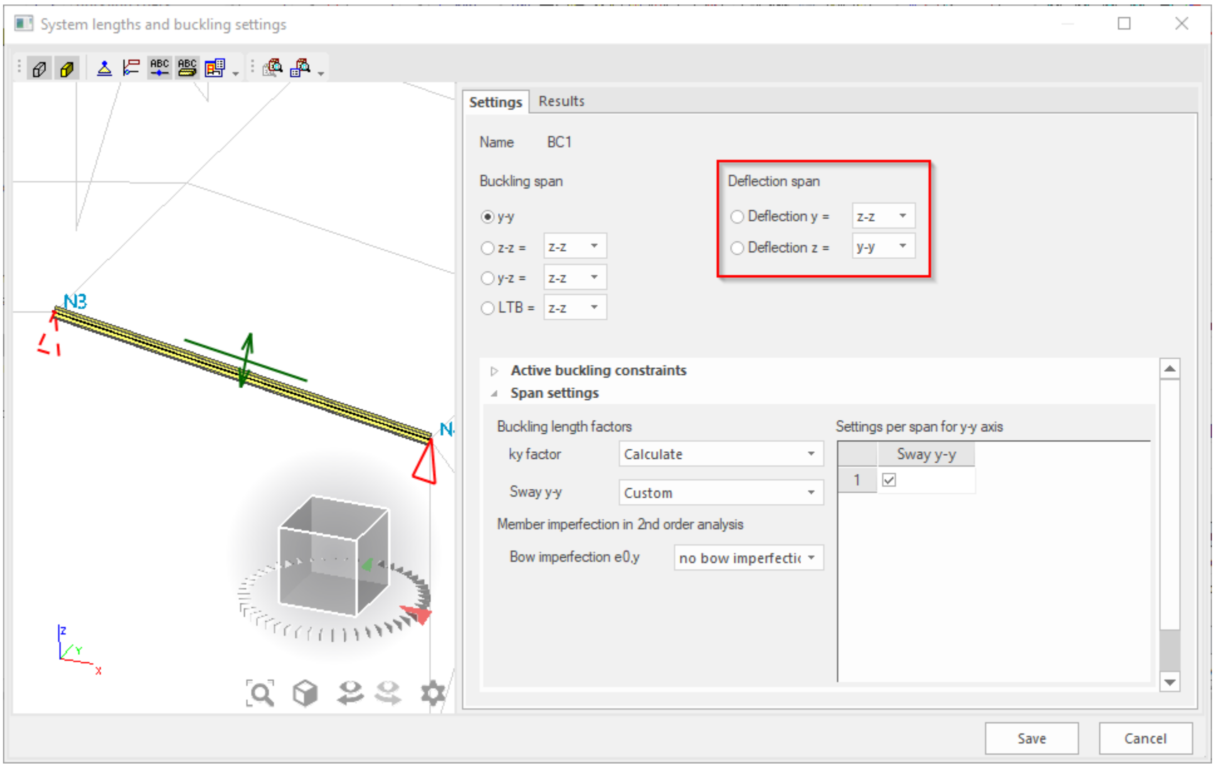


With the option “Steel -> Beams -> SLS Checks - Relative deformation”, the relative deformations can be checked. The relative deformations are given as absolute value, relative value related to the span, or as unity check related to the limit for the relative value to the span.

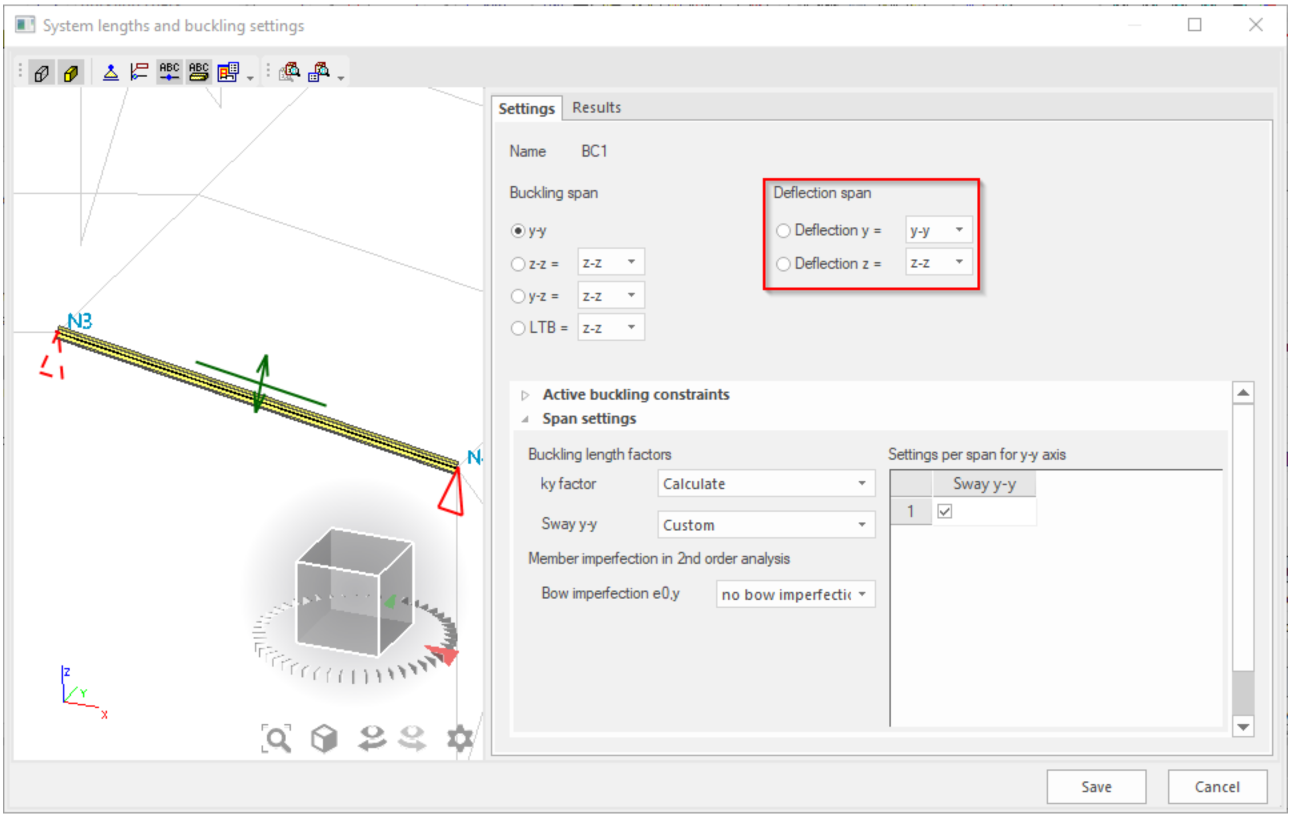
The span is defined in the menu “System lengths and buckling settings”, and is the distance between two buckling supports.

Select beam B3.

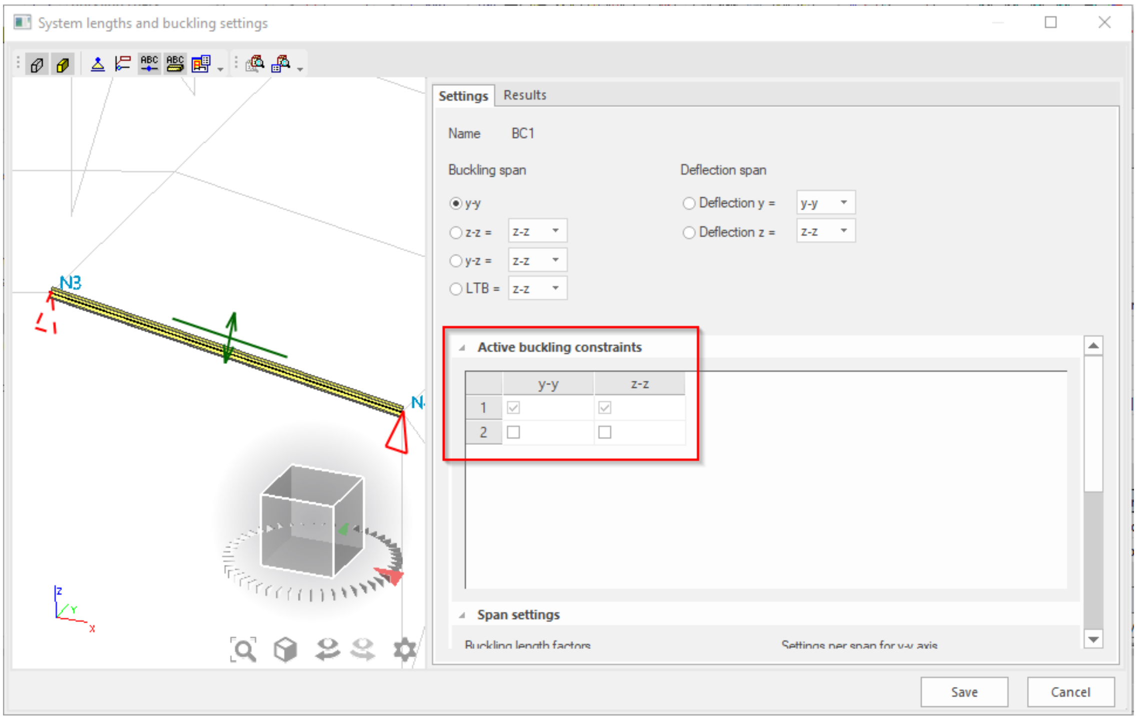
For the ‘systems’ of the relative deformation, by default a reference is made to the buckling system yy and zz.



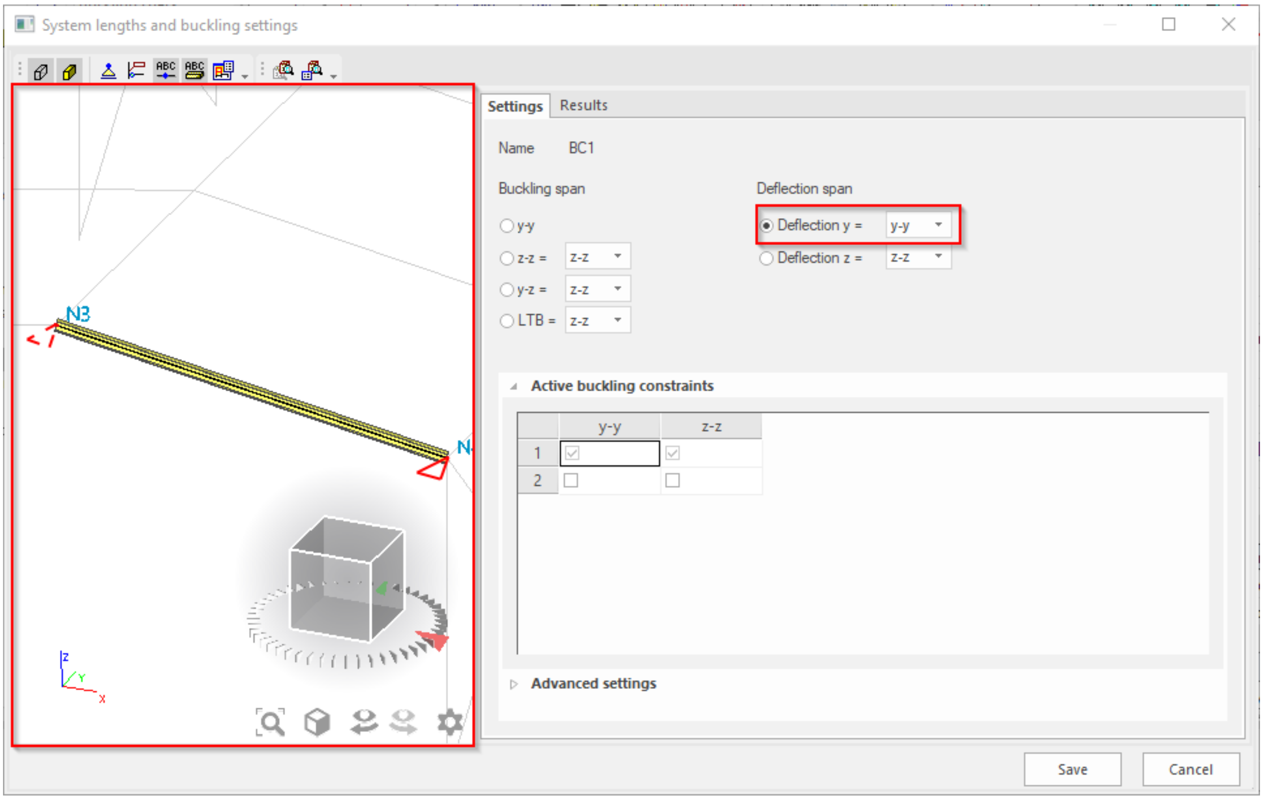
This can be changed to create a separate system for the relative deformation.



In the tab 'Active buckling constraints, the nodes can be fixed or free for the relative deformation.



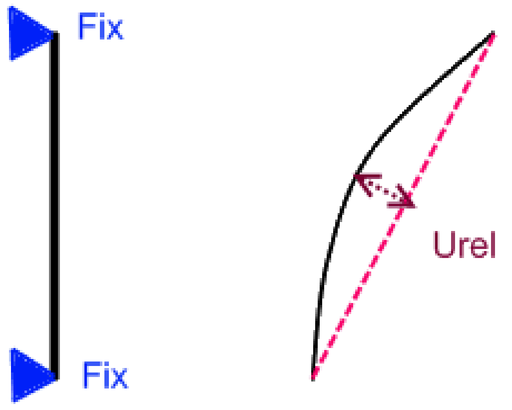
The same can be done using the graphical window, for both spans, y-y and z-z.



The relative deformation is always regarded according to the local axis of the member. For the deformation in the y direction of the member, there are two supporting nodes for member B3. The distance between these nodes is 6,083m.

- Relative deformation in the y direction (around the z axis).

  In this direction, there are 2 supported nodes. The deformation at the begin and end node is 0,0 mm. The maximal deformation for this beam is 11,1 mm.



Relative deformation

Linear calculation, Extreme : Global, System : Principal
Selection : B3
Combinations : CO1

Member	dx [m]	Case - combination	uy [mm]	Rel uy [1/xx]	uz [mm]	Rel uz [1/xx]	Check uy [-]	Check uz [-]
B3	6,083	CO1/1	**0,0**	**1/10000**	**-105,2**	**1/58**	**0,00**	**3,46**
B3	3,650	CO1/1	**11,1**	**1/547**	-75,9	1/80	**0,37**	2,49
B3	0,000	CO1/1	0,0	0	**0,0**	**0**	0,00	**0,00**

Length = 6,083 m

Deformation uy = 11,1 mm (and 0 mm at the beginning of the beam and 0 mm at the end of the beam)

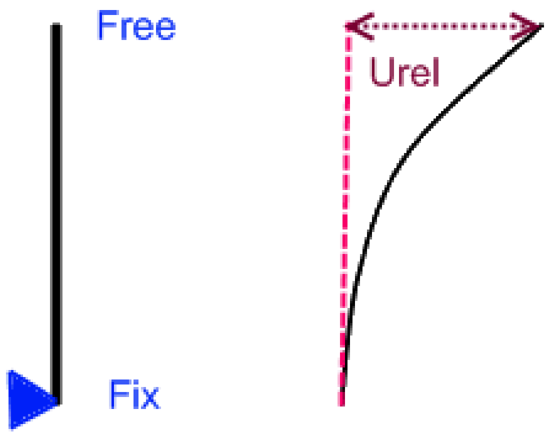
$uy_{relative}$ = 11,1 mm – 0.0 mm = 11,1 mm

$uy_{relative}$ / Length = 11,1 mm / 6083 mm = 1/547

Check = $\frac{1/547}{1/200} = 0{,}37$

- Relative deformation in the z direction (around the y axis).

  In this direction, there is one supported node. The deformation at the begin node is 0,0 mm and at the end node is 105,2 mm.



Length = 6,083 m

Deformation uz = 105,2 mm (and 0 mm at the beginning of the beam)

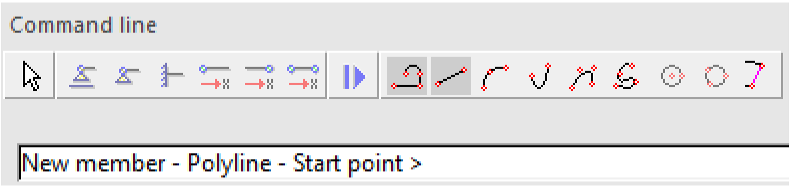
$uz_{relative}$ = 105,2 mm – 0.0 mm = 105,2 mm

$uz_{relative}$ / Length = 105,2 mm / 6083 mm = 1/58

$$\text{Check} = \frac{1/58}{1/200} = 3{,}46$$

The relative deformation of these members is now calculated with the length of the beam (distance between the buckling nodes). It is also possible to perform the relative deformation check for the complete span of the roof. This can be done by using polylines.

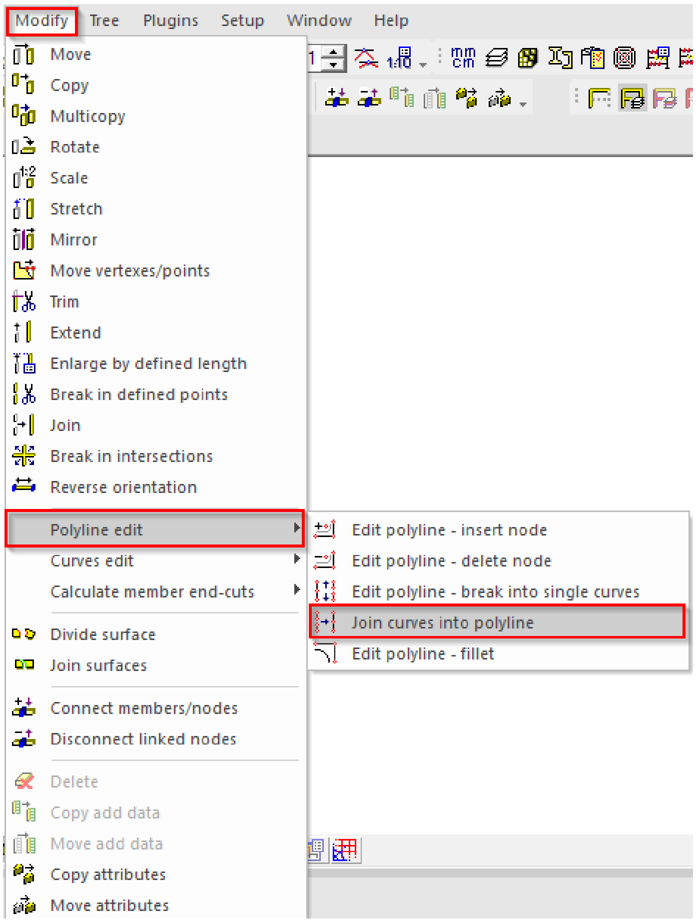
- Draw the members as polylines.



This can only be done if the elements are modeled using the option **Member** in the Structure menu.

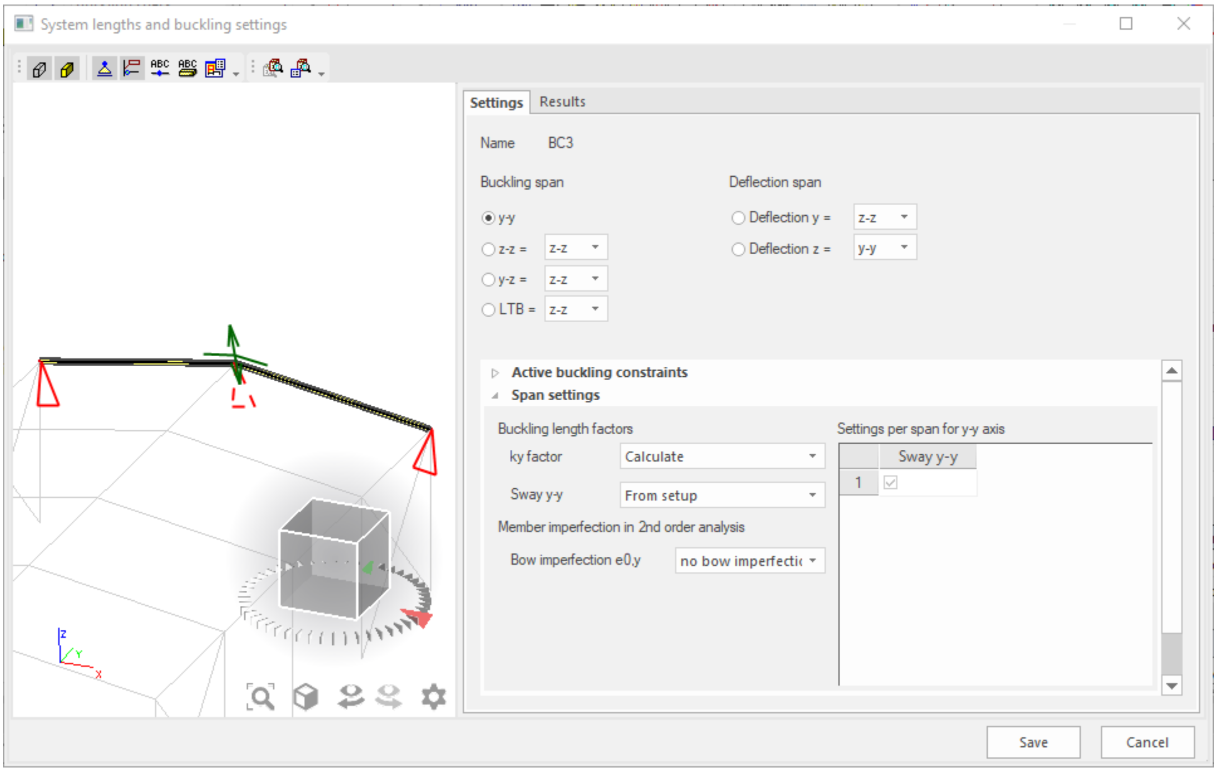
- Merging elements to a polyline.

Select beam B17 and B18. Through the option ‘Join curves into polyline’, these 2 beams will become 1 beam.



Since this has an influence on the results, the project needs to be recalculated.

By looking at the buckling system of the new member B18, it is clear that the two elements are taken as one for buckling and relative deformation.



This results in the following check for the relative deformation.

Relative deformation

Linear calculation, Extreme : Global, System : Principal
Selection : B18
Combinations : CO1

Member	dx [m]	Case - combination	uy [mm]	Rel uy [1/xx]	uz [mm]	Rel uz [1/xx]	Check uy [-]	Check uz [-]
B18	8,516	CO1/1	**-11,7**	**1/519**	-75,9	1/160	**0,39**	1,25
B18	0,000	CO1/1	**0,0**	**0**	**0,0**	**0**	**0,00**	**0,00**
B18	6,083	CO1/1	0,0	0	**-105,2**	**1/116**	0,00	**1,73**

References and literature

[1] Eurocode 3
Design of steel structures
Part 1 - 1 : General rules and rules for buildings
EN 1993-1-1:2005, 2005