SCIA scientific software

SCIA • ESA PT

2007

Engineering Structural Applications Professional Technology

Enabling Innovation in Construction

Manual

Steel Code Check

Steel Code Check

Table Of Contents

Steel Code Checks

Welcome

Thank you for choosing SCIA.ESA PT.

SCIA 1974-2007	**Version 2007.1**

scia		scia
	Steel Code Check	
scia		scia

Module Steel Code Check has been designed to facilitate the often-demanding task of design and checking of steel frame structures.

You can find more about the company and its products on www.scia-online.com.

Introduction to code checks

The ESA-Prima Win Steel Code Check module is a powerful program for the design of steel structures. It consists of stress and stability verifications of steel members according to a specific national code. It is also possible to search interactively for the lightest section, which meets the code requirements for selected loadings (optimisation).

The following structural steel design codes are supported

- EN 1993-1-8
- DIN 18800
- ONORM 4300
- NEN 6770-6771
- AISC - ASD: Allowable Stress Design
- AISC - LRFD: Load and Resistance Factor Design
- CM66
- BS5950
- SIA263

For more details about the used codes and the theoretical background, we refer to the SCIA.ESA PT Steel Code Check Theoretical Background.

This manual is written for all codes. The general screen copies are taken from the EC3 Code Check. Where the manipulation or input is different from this general way, a proper description and screen copy are given for the relevant code.

Parameters and settings for code checks

Basic parameters

Code independent basic parameters

Buckling length ratios ky, kz

Values of length ratios

There are various options for buckling ratios:

Inputted	The program always uses the input values.
Calculated	The program uses the calculated ky and kz factors and neglect all input values.
Calculated only if no inputted value	The program uses the calculated ky and kz factors only if there are no manually inputted values.
Bigger of inputted and calculated value	The program compares the inputted and calculated value for ky and kz factors and uses the greater value.
Lower of inputted and calculated value	The program compares the inputted and calculated value for ky and kz factors and uses the lower value.

Max. k ratio

The calculated value of k is limited and must not exceed the given value.

Max. Slenderness

If the slenderness of the checked member exceeds this value, the program prints a warning in the output report.

2^{nd} order buckling ratios

For 2^{nd} Order calculation, the buckling data as defined can be used, or the structure is considered as non-sway for all buckling data.

Note: For VARH members (see Steel Code Check Theoretical Background, Calculation of critical Euler force for VARH elements), only the first three options (Inputted, calculated, calculated only if no inputted value) are considered. In other cases, the critical Euler force is calculated.

Default sway types

These default sway types are used for all members, unless the user changes them in the settings made for particular members.

The sway type (with or without bracing) is used for calculation of buckling length ratios. We refer to Steel Code Check Theoretical Background, Calculation of buckling ratios for more information on this subject.

Elastic check only

If this option is checked, all members are assessed to elastic check only and no shear buckling check is performed.

Note: For EC 3,NEN6770/6771,BS5950, SIA263: check as class 3 section, Wel is used and no shear buckling is considered.

Section check only

If this option is checked, only the section check is carried out. No stability check is performed.

Check bounds

The unity checks are classified into three classes:

not-utilised	Unity check is lower than the Lower bound (the diagram employs purple colour).
optimal	Unity check is between Upper and Lower bound (the diagram employs green colour).
non-satisfying	Unity check is greater than the Upper bound (the diagram employs red colour).

In the **Check bounds** group the user may change the **Upper** and **Lower** bound. The default settings for the bounds are 0.25 for **Lower**, and 1 for **Upper**.

Limit for torsional check

This parameter introduces a limit for torsional check. This value refers to the unity check, based on the torsional shear stress only. If this value is exceeded, a composed elastic stress check (normal stress and shear stress check) is automatically performed regardless of the section classification.

Basic parameters for EC3

Safety factors

gamma M0	partial safety factor for resistance of Class 1, 2 or 3 cross-section (1.1)
gamma M1	partial safety factor for resistance of Class 4 cross-section (1.1) partial safety factor for resistance of member to buckling (1.1)
gamma M2	partial safety factor for resistance of net section at bolt holes (1.25)

ky, kz acc. to FEM.10.2.02.

If this option is selected the factors ky and kz are set according to FEM.10.2.02 (The Design of Steel Static Pallet Racking) regulations: during the stability check for combined bending and axial load, ky, kz = 1 if the unity check comes as a result of a second-order analysis.

Basic parameters for NEN 6770-6771

Safety factors

gamma M	partial safety factor (1.0)

LTB stiffeners only for lkip

When this option is ON, the LTB (lateral-torsional bucking) stiffeners will only influence the length of lkip. The value of lg is taken from buckling data.

m1, m2, mmid in buckling plane

When this option is selected, the values of moments My;1;s;d, My;2;s;d and My;mid;s;d are considered relative to the system length for buckling around the yy axis, and the values of moments Mz;1;s;d, Mz;2;s;d and Mz;mid;s;d are considered relative to the system length for buckling around the zz axis.

If this option is not selected, these values are considered relative to the member length.

Basic parameters for CSN 73 1401

Safety factors

gamma M0	partial safety factor for resistance of Class 1, 2 or 3 cross-section (1.15)
gamma M1	partial safety factor for resistance of Class 4 cross-section (1.15) partial safety factor for buckling resistance (1.15)
gamma M2	partial safety factor for resistance of net section (i.e. without bolt holes) (1.15)

Basic parameters for DIN 18800

Critical LTB according to ..

The critical moment for LTB can be calculated according to DIN 18800 (formula 19), according to EC3 - Annex F, or according to Roik, Carl and Lindner.

This selection is valid for symmetrical I shapes only.

For more information about the calculation of the critical LTB moment, please refer to Steel Code Check Theoretical Background, DIN 18800.

Double bending check with ...

For double bending, either checking method 1 (formula 28) or method 2 (formula 29) can be chosen.

LTB check for RHS/CHS sections

The LTB check can be activated for RHS / CHS sections (Rectangular Hollow Section / Circular Hollow Section).

Only LTB stability check in 2nd Order calculation

When this option is selected, there is no buckling check performed. Only LTB check is carried out in the stability check.

Elastic check according to T1 article 750

With this option is ON, the potential plastification described in the stated article, is used.

Basic parameters for ÖNORM B 4300

Critical LTB according to ..

The critical moment for LTB can be calculated according to DIN 18800 (formula 19), according to EC3 - Annex F, or according to Roik, Carl and Lindner. This selection is only valid for symmetrical I shapes. For more information about the calculation of the critical LTB moment, we refer to Steel Code Check Theoretical Background, DIN 18800.

Double bending check with ...

For double bending, the check method 1 (formula 28) or method 2 (formula 29) can be activated.

LTB check for RHS/CHS sections

The LTB check can be activated for RHS / CHS sections (Rectangular Hollow Section / Circular Hollow Section).

Only LTB stability check in 2nd Order calculation

When this option is selected, there is no buckling check performed. Only LTB check during the stability check is carried out.

Elastic check according to T1 article 750

When this option is ON, the potential plastification described in the mentioned article, will be used.

Note: Basic parameters for ÖNORM B 4300 are the same as for DIN 18800.

Basic parameters for BS5950

Section check based on

For plastic and compact sections, BS5950 Art. 4.8.2. & 4.8.3.2. (b) prescribes a detailed approach to determine the unity check of axially loaded members with moments. The detailed relationship allows greater economy for plastic and compact section. In this expression, we use a reduced moment capacity Mr about the major and the minor axis respectively. Those values are determined by means of EC3 Art.5.4.9. For semi-compact and slender section, the simplified approach is applied following Art. 4.8.2.and Art. 4.8.3.2. (a). It is possible to choose the approach used to perform the control. In the case of a class3 or 4 section, even if the user has selected the EC3 method, an elastic approach following BS interaction will be performed.

Calculation type of m, n

m represents the equivalent uniform moment distribution and **n** represents the equivalent slenderness factor. Those factors are necessary to perform the lateral-torsional buckling check. They are determined as follows:

- For a beam without loading point between points of lateral restraint, n=1 and m depends on the ratio of the end moments at the points of restraint.
- For a beam loaded between points of lateral restraint, m=1 and n depends on the ratio of the end moments at the points of restraint and on the ratio of the larger moment to the mid-span free moment.

There are thus two methods for dealing with lateral-torsional buckling namely:

- 'm approach' i.e. the 'equivalent uniform moment method' with n=1

- 'n approach' i.e. the 'equivalent slenderness method' with m=1

In any given situation, only one method will be admissible, taking into account that it is always conservative to use m=n=1. In the Calculation of m, n group, you can let the program to determine both values (m method or n method will be automatically determined), choose the 'm method' or select the safe solution m=n=1.

Basic parameters for CM66

Additif 80

Check this box in order to perform the check according to Additif 80 for symmetrical I sections and RHS sections. Other sections will be checked according the elastic rules from CM66.

Fictive slenderness only for k1

Check this box if you want to take the fictive slenderness into account only for the value of k1x and k1y.

If the option is not checked, the fictive slenderness will influence the values of kfx, kfy, k1x and k1y.

Basic parameters for GBJ 17-88

Activate plastic check

The plastic check can be activated, if the section fulfils the plastic conditions.

Basic parameters for BSK99

Critical LTB according to ..

The critical moment for LTB can be calculated according to BSK99 or according to EC3 - Annex F. This selection is only valid for symmetrical I shapes.

For more information about the calculation of the critical LTB moment, we refer to Steel Code Check Theoretical Background, BSK99.

Basic parameters for SIA263

Safety factors

Gamma M1	partial safety factor for resistance
Gamma M2	partial safety factor for resistance of net section

Buckling parameters

Default buckling parameters

Description

The default buckling parameters are used whenever a new beam is input into your project. By default, the new beam takes these default parameters. If required, you may later alter these default values and assign a specific values to the particular beam.

zz	System length for buckling around the local zz axis (weak axis). This is usually the length between the points braced in the direction of the local yy axis.
yz	System length for torsional buckling. This is the length between the restraints for torsion. Both EC3, DIN18800, ONORM4300, NEN6770, SIA263, AISC-ASD and AISC-LRFD take account of torsional buckling.
lt	System length for lateral-torsional buckling. This is usually the length between the points braced in yy direction (= length between the lateral restraints).
def yy	System length for deformation around the local yy axis (strong axis).
def zz	System length for deformation around the local zz axis (weak axis).
ky	The program uses either the inputted or calculated values.

kz	Alternatively, the buckling length can be input directly. For a theoretical explanation about the calculation of buckling ratios ky and kz, see Steel Code Check Theoretical Background, Calculation of buckling ratios.
Influence of load position	This field is relevant for lateral-torsional buckling check. It provides for consideration of destabilising loads in moment factors for LTB (See Steel Code Check Theoretical Background, Calculation of moment factors for LTB). Destabilising loads are loads that act above the level of the beam's shear centre and are free to move sideways with the beam as it buckles (and produce a disturbing effect)

Procedure

The procedure is described in chapter Adjusting the default buckling parameters. For your convenience, it is repeated here as well.

The procedure for adjustment of default buckling parameters

1. Open service **Steel**:
   a. either using tree menu function **Steel**,
   b. or using menu function **Tree > Steel**.
2. Select function **Setup** and open it.
3. In the dialogue that appears on the screen, select tab **Setup of buckling**.
4. Type required values and select appropriate options.
5. Confirm with **[OK]**.

Note: For the relation between default parameters and parameters defined for a particular member read also chapter Principle of input of buckling parameters .

Buckling parameters for a particular beam

Description

When a new beam is defined (i.e. inserted into your model of the analysed structure), it reads the default buckling parameters adjusted through function **Setup** in service **Steel**.

In practice however, it may be necessary to adjust the parameters differently for certain beams. This can be done in the adjustment dialogue for a particular member.

In addition, this adjustment offers some additional options next to those available in the setup of default parameters.

Buckling and relative lengths dialogue

The **Buckling and relative lengths dialogue** is divided into two parts:

- input for the whole member – base settings,
- input for individual intervals of the member – buckling data.

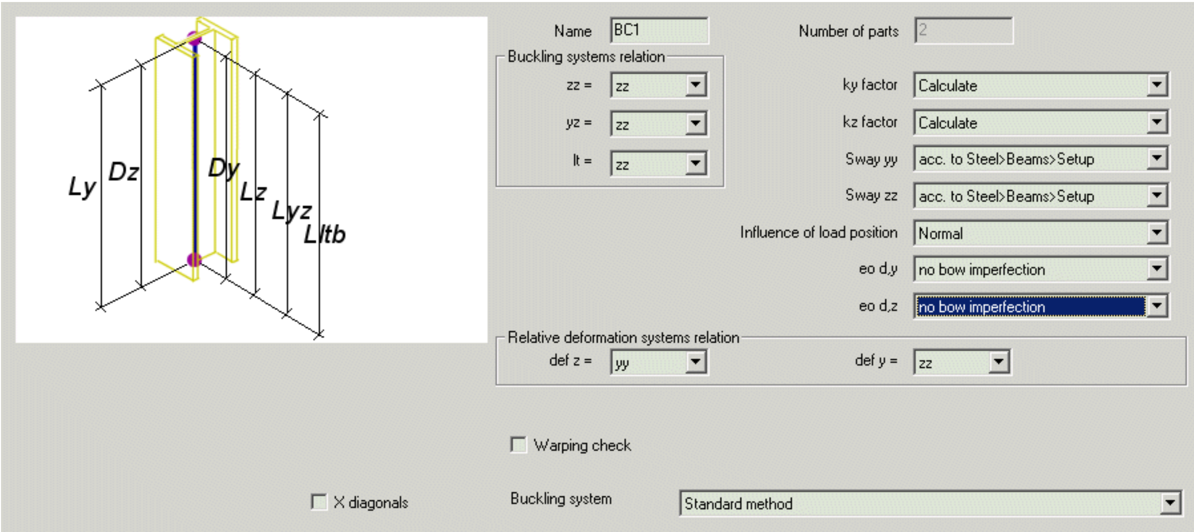
Base settings

Name	Identifies the buckling length system
Number of parts	(informative) Show the number of intervals of the system.
Buckling systems relation: zz	System length for buckling around the local zz axis (weak axis). This is usually the length between the points braced in the direction of the local yy axis. Alternatively, it is possible to define a separate zz system.
Buckling systems relation: yz	System length for torsional buckling. This is the length between the restraints for torsion. This system may be defined separately, or it may be said that it is identical with yy or zz system.
Buckling systems relation: lt	System length for lateral-torsional buckling. This is usually the length between the points braced in yy direction (= length between the lateral restraints). Alternatively, it is possible to define

	a separate lt system or make it identical with zz system.
Relative deformation systems relation: **def z;** **def y**	System length for deformation. The same rules as for **Buckling systems relation** apply here.
ky factor; **kz factor**	The program uses either the inputted or calculated values. Alternatively, the buckling length can be input directly. For a theoretical explanation about the calculation of buckling ratios ky and kz, see Steel Code Check Theoretical Background, Calculation of buckling ratios.
sway yy; **sway zz;** **X diagonals**	The sway type is used to determine the relevant buckling factor k. If the box is checked, the member is sway in this direction. It is possible to say directly YES or NO, or to say that the values is to be taken from the default parameters. Example Sway y-y is set to YES: the member is sway in the plane perpendicular to the local YY axis of the member (the local ZZ plane) - Sway for buckling around the local YY axis. If the **X diagonal** check box is checked, the buckling factor is calculated according to DIN18800 Teil 2, Table15 (see Steel Code Check Theoretical Background, Calculation of buckling ratio) on condition that member satisfies the specifications given in the chapter. **Note**: If a 2^{nd} -order calculation is performed, the buckling factors are calculated in the non-sway mode or in the defined mode, depending on the settings in the default data.
Influence of load position	This field is relevant for lateral-torsional buckling check. It provides for consideration of destabilising loads in moment factors for LTB (See Steel Code Check Theoretical Background, Calculation of moment factors for LTB). Destabilising loads are loads that act above the level of the beam's shear centre and are free to move sideways with the beam as it buckles (and produce a disturbing effect)
eo d,y; **eo d,z**	The following code-based options are available: • bow imperfection according to code – elastic (the bow imperfection is calculated according to the code specifications), • bow imperfection according to code – plastic (the bow imperfection is calculated according to the code specifications), • bow imperfection according to code - elastic - only if required (the bow imperfection is calculated according to the code specifications, if required by the normal force condition), • bow imperfection according to code - plastic - only if required (the bow imperfection is calculated according to the code specifications, if required by the normal force condition), • no bow imperfection.
Buckling system	The following code-based options are available: • standard method, • crossing diagonals (to DIN), • leg with symmetrical bracing,

- leg with intermediate transverse support,
- leg with staggered bracing,
- single bracing with SBS,
- cross bracing,
- K bracing,
- horizontal bracing,
- discontinuous cross bracing with horizontal member.

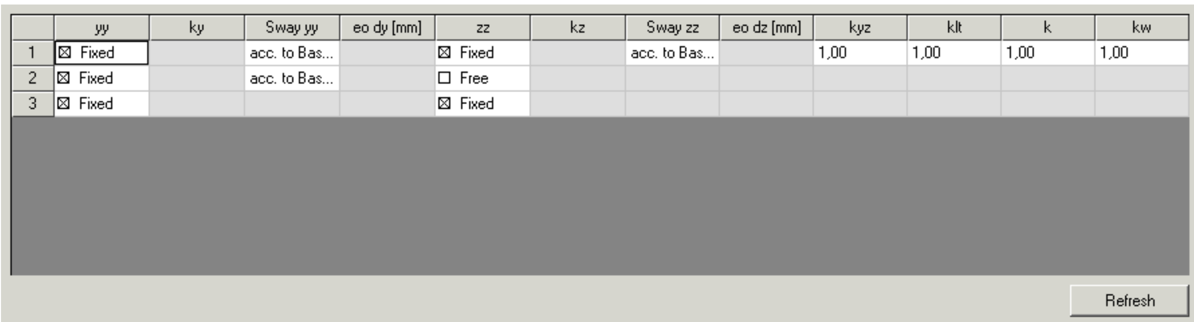
The individual options are explained in the theoretical background for Steel Code Check. See the chapters: (i) Calculation of buckling ratio > calculation of buckling ratio - general formula; (ii) Calculation of buckling ratio > calculation of buckling ratio for crossing diagonals; (iii) Calculation of buckling ratio > calculation of buckling ratio for lattice tower members.



Buckling data

Depending on the adjustments made in Base settings (see above), some of the parameters for individual intervals of the buckling length system may be specified in the second part of the dialogue.

Some of the parameters contained in this dialogue are code-dependent and appear only for the appropriate national technical code.



Note: For the relation between default parameters and parameters defined for a particular member read also chapter Principle of input of buckling parameters .

Procedure

The procedure is described in chapter Adjusting the buckling parameters for particular members. For your convenience, it is repeated here.

The procedure for adjustment of buckling parameters for a particular member

1. In the graphical window, select the beam (or beams) whose buckling settings should be modified.

2. The beam properties are displayed in the Property window.
3. In the table cell **System lengths** use the combo box to select the required **System length definition** and go to the last step of the procedure.
4. If the required **System length definition** has not been defined yet, use the button at the right hand side of the cell to create a new **System length definition**.
5. Press button **Edit buckling** to open the editing dialogue.
6. Adjust required parameters.
7. Confirm with **[OK]**.
8. Clear the selection.

Principle of input of buckling parameters

SCIA.ESA PT uses for some functions a hierarchical order: application – function / system – a single part.

The same principle is used for setting of buckling parameters.

The whole application

The parameters may be set for the whole application. The parameters are valid for the whole structure if nothing else is set otherwise.

The system

If necessary, the global settings may be altered for a particular member. The member may consist of several "buckling" intervals. These intervals together form a "buckling system" and such a system may be tuned by means of explicitly defined parameters.

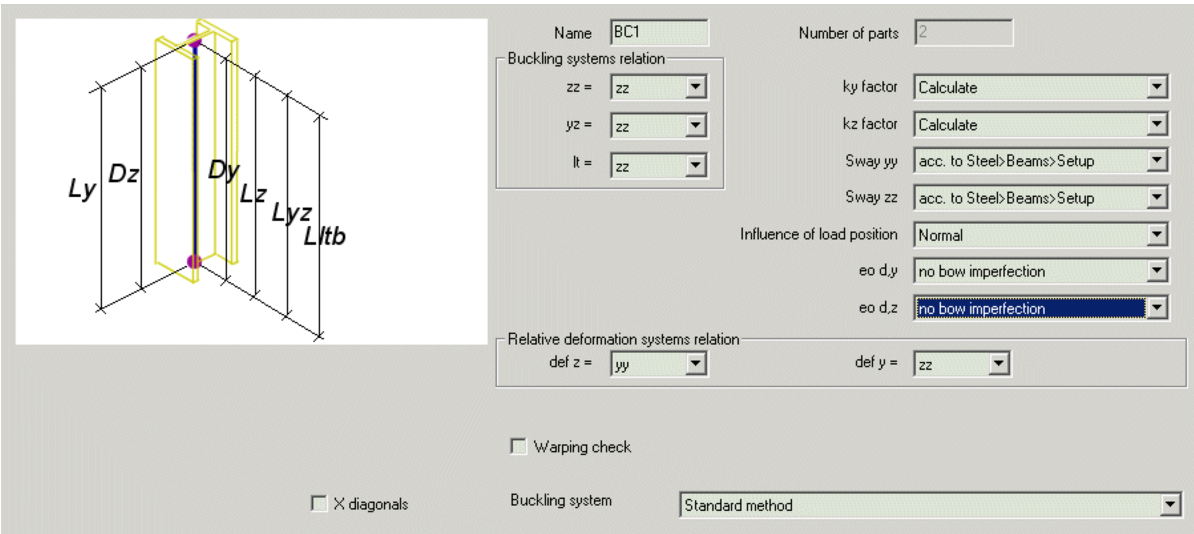
The part

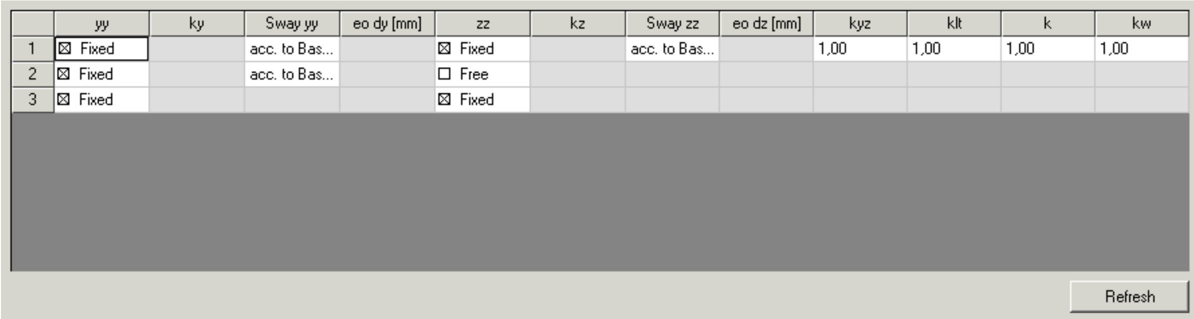
As stated above, the "buckling system" of a beam may consist of several intervals. And different parameters may be assigned to each of the intervals.

Practical application of the Hierarchical Principle in ESA

The global buckling settings valid for the whole application can be set via function Steel > Setup > Setup of buckling.

The parameters of the "buckling system" of a particular beam and also the parameters for individual intervals can be adjusted in the Bucking data dialogue (see below).





	yy	ky	Sway yy	eo dy [mm]	zz	kz	Sway zz	eo dz [mm]	kyz	klt	k	kw
1	☒ Fixed		acc. to Bas...		☒ Fixed		acc. to Bas...		1,00	1,00	1,00	1,00
2	☒ Fixed		acc. to Bas...		☐ Free							
3	☒ Fixed				☒ Fixed							

Let's have a look at **Sway** settings for both **yy** and **zz** direction. The **Sway** parameter can be adjusted twice in the **Buckling data dialogue**. One setting can be made for the whole "buckling system" of the selected beam or beams (the setting at the right-hand side of the dialogue). The other setting can be made in the table at the bottom part of the dialogue and is valid for a particular buckling interval.

System value	**Interval value**	**Calculation value**
the value set in "buckling system" settings (the right-hand side of the dialogue)	the value set in "buckling interval" settings (the bottom part of the dialogue)	the value used for calculations
Sway yy = Settings	Sway yy = Settings	The global value adjusted in Steel > Setup > Setup of buckling will be used. System value accepts the global setting. Interval value accepts the system setting.
Sway yy = Settings	Sway yy = Yes	Sway yy = YES will be used. System value accepts the global setting. Interval value overwrites the system setting.
Sway yy = Settings	Sway yy = No	Sway yy = NO will be used. System value accepts the global setting. Interval value overwrites the system setting.
Sway zz = Yes	Sway zz = Settings	Sway zz = YES will be used. System value overwrites the global setting. Interval value accepts the system setting.
Sway zz = Yes	Sway zz = Yes	Sway zz = YES will be used. System value overwrites the global setting. Interval value overwrites the system setting.
Sway zz = Yes	Sway zz = No	Sway zz = NO will be used. System value overwrites the global setting. Interval value overwrites the system setting.

Buckling parameters for EC3

Warping check

If this option is ON, a warping check is performed on the member. The end conditions for warping (free, fixed) can be set separately for both ends of member. The implementation of the warping check is described in Steel Code Check Theoretical Background, Warping check.

k factor for lateral-torsional buckling

k	Refers to rotational end-restraint 'in plan' (around the local zz axis). For end-conditions given greater restrained to rotation in plan, values of k < 1.0 can be used. The value can vary from 0.5 for full fixity, to 1.0 for no fixity, with 0.7 for one end fixed and one end free.
kw	Refers to end warping (EC3, Annex F).Unless special provision for warping

	fixity is made, kw should be taken as 1.0.The presence of endplates will influence this value.

Note: The lateral-torsional buckling check uses the formulas for Mcr (the elastic critical moment) as defined in EC3-Annex F. When the cross section type is not supported by this Annex F (e.g. U sections) , a general formula is used, where it is supposed that the applied loadings are passing through the shear centre of the section. More info is given in the 'Theoretical Background'.

Buckling parameters for NEN 6770-6771

Fy; tot; s; d and Fz; tot; s; d

For code NEN 6770 / 6771, value Fy;tot;s;d and Fz;tot;s;d can be modified with the so-called "aanpendelende belasting" in ly and lz section as follows:

$$F; tot; s; d = Nc; s; d + Nc; s; d \times A + B$$

The factor A and the value B can be introduced.

k factor for lateral-torsional buckling

kLTB	The lateral torsional buckling (LTB) length is defined by lLTB = kLTB * LLTB
k1	The length lkip is given by lkip = lLTB * k1
kg	The length lg is given by lg = lLTB * kg

Warping check

If this option is ON, a warping check is performed on the member. The end conditions for warping (free, fixed) can be set separately for both ends of member. The implementation of the warping check is described in Steel Code Check Theoretical Background, Warping check.

Buckling parameters for CSN 73 1401

Other lateral-torsional buckling parameters

kappa M	If this parameter is NOT equal to zero and, at the same time, both Kappa 1 and Kappa 2 ARE equal to zero, the calculation of slenderness for lateral-torsional buckling is carried out in accordance with Appendix H6 of the code.
Kappa 1	If this parameter is NOT equal to zero and simultaneously Kappa 2 is NOT equal to zero, the calculation of slenderness for lateral-torsional buckling is carried out in accordance with Appendix H2 of the code.
Kappa 2	If this parameter is NOT equal to zero and simultaneously Kappa 1 is NOT equal to zero, the calculation of slenderness for lateral-torsional buckling is carried out in accordance with Appendix H2 of the code.
lambda lt (**lam_lt)**	Direct input of slenderness for lateral-torsional buckling. If a non-zero value is given, the value is always considered as the slenderness for lateral-torsional buckling. ATTENTION: This parameter must ALWAYS be input for U-profiles (see below)!

Important note: Lateral-torsional buckling check for U-profiles gives a completely misleading result if parameter **lambda lt** is not defined.

Buckling parameters for DIN 18800

Warping check

If this option is ON, a warping check is performed on the member. The end conditions for warping (free, fixed) can be set separately for both ends of member. The implementation of the warping check is described in Steel Code Check Theoretical Background, Warping check.

ßz	This parameter refers to rotational end-restraint ‘in plan’ (around the local zz axis). For end-conditions given greater restrained to rotation in plan, values of ß < 1.0 can be used. The value can vary from 0.5 for full fixity, to 1.0 for no

	fixity, with 0.7 for one end fixed and one end free.
ß0	This refers to end warping. Unless special provision for warping fixity is made, ß0 should be taken equal to 1.0.The presence of endplates will influence this value.

Buckling parameters for ÖNorm B 4300

Warping check

If this option is ON, a warping check is performed on the member. The end conditions for warping (free, fixed) can be set separately for both ends of member. The implementation of the warping check is described in Steel Code Check Theoretical Background, Warping check.

ßz	This parameter refers to rotational end-restraint 'in plan' (around the local zz axis). For end-conditions given greater restrained to rotation in plan, values of ß < 1.0 can be used. The value can vary from 0.5 for full fixity, to 1.0 for no fixity, with 0.7 for one end fixed and one end free.
ß0	This refers to end warping. Unless special provision for warping fixity is made, ß0 should be taken equal to 1.0.The presence of endplates will influence this value.

Note: Basic parameters for ÖNORM B 4300 are the same as for DIN 18800.

Buckling parameters for CM 66

k factor for lateral-torsional buckling

kLTB	The lateral-torsional buckling (LTB) length is defined by lLTB = kLTB * LLTB. This is the equivalent of length l0 in CM66: the length between supports for LTB.
k1	The length l is given by l = lLTB * k. l is defined in CM66 as the buckling length of the compressed part that is presumed to be isolated from the element.

Buckling parameters for SIA263

Warping check

If this option is ON, a warping check is performed on the member. The end conditions for warping (free, fixed) can be set separately for both ends of member. The implementation of the warping check is described in Steel Code Check Theoretical Background, Warping check.

k factor for lateral-torsional buckling

k	Refers to rotational end-restraint 'in plan' (around the local zz axis). For end-conditions given greater restrained to rotation in plan, values of k < 1.0 can be used. The value can vary from 0.5 for full fixity, to 1.0 for no fixity, with 0.7 for one end fixed and one end free.
kw	Refers to end warping (EC3, Annex F).Unless special provision for warping fixity is made, kw should be taken as 1.0.The presence of endplates will influence this value.

Note: The lateral-torsional buckling check uses the formulas for Mcr (the elastic critical moment) as defined in SIA263. When the cross section type is not supported in SIA263, the formulas from EC3-Annex F, or a general formula (e.g. U sections) used. In this latest formula , it is supposed that the applied loadings are passing through the shear centre of the section. More info is given in the 'Theoretical Background'

Deflection parameters

Permissible relative deformation

The maximum permissible relative deformation may be adjusted separately for individual beam types:

- general beam,
- beam,
- column,
- gable column,
- secondary column,
- rafter,
- purlin,
- roof bracing,
- wall bracing,
- girt,
- truss chord,
- truss diagonal,
- plate rib.

Fire resistance parameters

General fire resistance parameters for EC3

The general parameters define applied procedures and specify code-related parameters used throughout the checking of a beam concerning its fire resistance:

temperature curve	Available temperature curves are: • ISO 834 curve • external fire curve • hydrocarbon curve • smouldering fire.
coefficient of heat transfer by convection	Default value is 25 W/m²K α_c - ENV 1991-2-2 Art. 4.1.(8)
emissivity related to fire compartment	Default value is 0.8. ε_l – ENV 1991-2-2 Art. 4.2.1.(3)
emissivity related to surface material	Default value is 0.625. ε_m - ENV 1991-2-2 Art. 4.2.1.(3)
correction factor for beam exposed on 3 sides	Adaptation factor for non-uniform temperature distribution across a cross section exposed on three sides. Default value = 0.70. K_1 - ENV 1993-1-2:, 4.2.3.3. (8).
correction factor for columns and beams	This value is a correction factor that allows for a number of effects, including the difference in the strain at failure. The value is empirical. Default value = 1.2 ENV 1993-1-2, 4.2.3.2. (1). ENV 1993-1-2, 4.2.3.3. (5).
factor for net heat flux	Default value is 1.0.

(convection part)	$\gamma_{n,c}$ - ENV 1991-2-2 Art. 4.2.1.(2)
factor for net heat flux (radiation part)	Default value is 1.0. $\gamma_{n,r}$ - ENV 1991-2-2 Art. 4.2.1.(2)
configuration factor for radiation heat flux	Default value is 1.0. Φ - ENV 1991-2-2 Art. 4.1.(4)
analysis type	The fire resistance check can be performed in three domains: • strength domain, • temperature domain, • time domain. In the resistance domain, the resistance is checked after the imposed time. In the temperature / time domain, the material temperature (after the imposed time) is checked in relation to the critical material temperature.
iterative process	The critical material temperature is calculated using the analytical formulas of the code, or by an iterative process.
model fire engineering	The Fire Resistance checking is carried out by means of checks defined in ENV 1993-1-2:1995 or by means of checks as they are defined in 'ECCS N° 111 - Model Code on Fire Engineering'.
safety factor for fire situation	The partial safety factor for fire situation, default value is 1.0 $\gamma_{m,fi}$ - ENV 1993-1-2 Art. 2.3.(1), (2), (3)

General fire resistance parameters for NEN 6072

The general parameters define applied procedures and specify code-related parameters used throughout the checking of a beam concerning its fire resistance:

temperature curve	Available temperature curves are: • ISO 834 curve, • external fire curve, • hydrocarbon curve, • smouldering fire.
correction factor for beam exposed on 3 sides	Adaptation factor for non-uniform temperature distribution across a cross section exposed on three sides. Default value = 0.70.
correction factor for columns and beams	This value is a correction factor that allows for a number of effects, including the difference in the strain at failure. The value is empirical. Default value = 1.2.
analysis type	The fire resistance check can be performed in three domains: • strength domain, • temperature domain, • time domain. In the resistance domain, the resistance is checked after the imposed time. In the temperature or time domain, the material temperature (after the imposed time) is checked in relation to the critical material temperature.

iterative process	The critical material temperature is calculated using the analytical formulas of the code, or by an iterative process.
safety factor for fire situation	The partial safety factor for fire situation. Default value is 1.0.

General fire resistance parameters for SIA263

The general parameters define applied procedures and specify code-related parameters used throughout the checking of a beam concerning its fire resistance:

temperature curve	Available temperature curves are: • ISO 834 curve • external fire curve • hydrocarbon curve • smouldering fire.
coefficient of heat transfer by convection	Default value is 25 W/m²K
emissivity related to fire compartment	Default value is 0.8.
emissivity related to surface material	Default value is 0.625.
correction factor for beam exposed on 3 sides	Adaptation factor for non-uniform temperature distribution across a cross section exposed on three sides. Default value = 0.70.
correction factor for columns and beams	This value is a correction factor that allows for a number of effects, including the difference in the strain at failure. The value is empirical. Default value = 1.2 See ENV 1993-1-2:1995, 4.2.3.2. (1).
factor for net heat flux (convection part)	Default value is 1.0.
factor for net heat flux (radiation part)	Default value is 1.0.
configuration factor for radiation heat flux	Default value is 1.0.
analysis type	The fire resistance check can be performed in three domains: • strength domain, • temperature domain, • time domain. In the resistance domain, the resistance is checked after the imposed time. In the temperature / time domain, the material temperature (after the imposed time) is checked in relation to the critical material temperature.
iterative process	The critical material temperature is calculated using the analytical formulas of the code, or by an iterative process.
safety factor for fire situation	The partial safety factor for fire situation, default value is 1.0

Member related fire resistance parameters

It is possible to set particular fire resistance parameters to individual members in the structure.

time resistance	Specifies the required fire resistance.

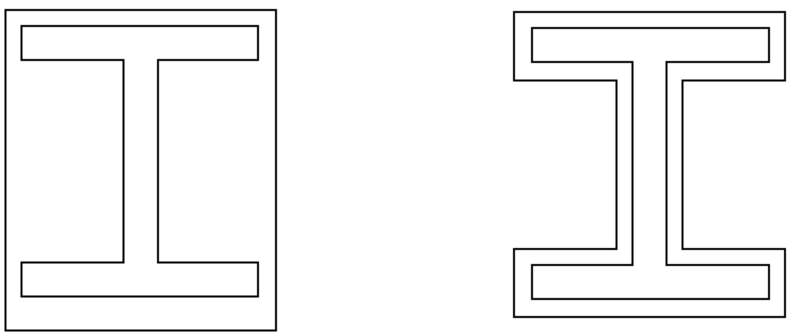
buckling ratio ky	When this ratio is greater than 0.0, the 'normal' defined buckling ratio ky will be overruled by this value.
buckling ratio kz	When this ratio is greater than 0.0, the 'normal' defined buckling ratio kz will be overruled by this value.
fire exposure	The section may exposed to fire on all or only three sides.
covered flange	When section is exposed to fire on 3 sides, the covered flange must be defined.
protection	Defines whether the section is protected or not.
insulation	Defines the type of insulation used as a protection.
thickness	Specifies the thickness of the protection.
k2 **Multiplication factor for correction**	This parameters is the adaptation factor for non-uniform temperature distribution along the beam. Recommended values : k2 = 0.85 : at supports of a statically indeterminate beam, k2 = 1.00 : for all other cases. ENV 1993-1-2 Art. 4.2.3.3.(9)

Insulation parameters

If a particular member in the structure is protected by a kind of insulation, the user may specify the parameters of such insulation.

Encasement type	Possible types are: hollow encasement, contour encasement (see picture below).
Insulation type	The insulation may be: board protection, spray protection, intumescent coating.
Unit mass	self-explanatory
Thermal conductivity	self-explanatory
Specific heat	self-explanatory
Default value for thickness	Each particular insulation type may be used in the structure several times. Each time it may be applied with a different thickness. The parameter here defines the default value of thickness. The concrete thickness for each particular application may be adjusted in the dialogue of Member related fire resistance parameters.
Kd,ef	Effective coefficient of heat transfer for intumescent coating.

Encasement type



hollow encasement	contour encasement

Member settings

Member settings

The user may define a set of member parameters for each beam in the structure and thus control the type of checking.

Name	Defines the name of member data set.
Section classification	This item allows the user to decide whether the classification of cross-section should be performed automatically by the program or whether the used himself/herself takes the responsibility of classification.
Elastic check only	If this option is ON, only the elastic check is carried out.
Section check only	If this option is ON, only the section check is carried out.
Field	This group of items provides for the definition of a "field" or interval where the adjusted data are valid.

Lateral torsional buckling restraints

Lateral-torsional buckling settings

LTB restraints are supports against lateral-torsional buckling (LTB) at the top or bottom flange of the beam. The topside is defined by the positive local z- axis of the section. It means that for a positive My (which causes compression at the topside) the LTB length (and the related moment factors - see Steel Code Check Theoretical Background, Calculation of moment factors for LTB) is calculated by the position of the stiffeners at the topside. The bottom side is defined by the negative local z-axis of the section. It means that for a negative moment My (which causes compression at the bottom side) the LTB length (and the related moment factors - see Steel Code Check Theoretical Background, Calculation of moment factors for LTB) is calculated by the position of the stiffeners at the bottom side. When no LTB stiffeners are defined, the values, introduced in the Buckling data dialog are used.

If required, it is possible to define the position of points where lateral-torsional buckling is prevented.

Name	Defines the name of restraint.
Position z	Specifies the position in Z direction, i.e. either the topside or the bottom side.
Position x	Specifies the position in X (longitudinal) direction.
Co-ordinate definition	Defines the co-ordinate system in which the **position x** is inputted.
Repeat	Tells how many times the restraint is repeated.
Delta x	Defines the distance between individual restraints. This item is available only if **Repeat** is greater than one.

Stiffeners

Stiffener settings

Web stiffeners are used to prevent shear buckling, which can be a design failure mode for high and slender (thin webs) profiles.

If required, it is possible to define the position and dimensions of stiffeners.

Name	Defines the name of the stiffener (set of stiffeners).

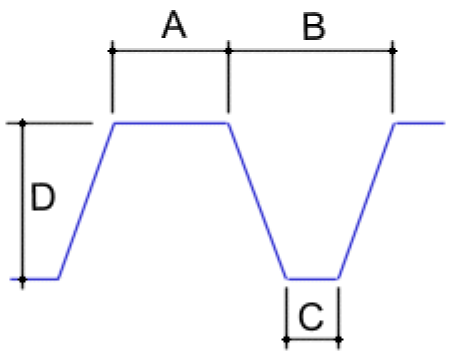
Material	Specifies the material used for the stiffener.
Thickness	Determines the thickness of the stiffener.
Decrease	
Position x	Specifies the position in X (longitudinal) direction.
Co-ordinate definition	Defines the co-ordinate system in which the **position x** is inputted.
Repeat	Tells how many times the restraint is repeated.
Regularly	Tells that stiffeners are positioned regularly with the same distance between two adjacent stiffeners.
Delta x	Defines the distance between individual stiffeners. This item is available only if **Repeat** is greater than one.
On begin	Specifies whether the first stiffener should be applied.
On end	Specifies whether the last stiffener should be applied.

Diaphragms

Diaphragm definition

A diaphragm is defined by a set of basic parameters.

Name	Name of the diaphragm.
I moment	Moment of inertia (i.e. second moment of area) per length.
K1 +	Factor K1 for positive position.
K2 +	Factor K2 for positive position.
K1 -	Factor K1 for negative position.
K2 -	Factor K2 for negative position.
A	Dimension A: see the Fig. below.
B	Dimension B: see the Fig. below.
C	Dimension C: see the Fig. below.
D	Dimension D: see the Fig. below.
Thickness	Thickness of the sheet.



In addition, other settings must be made in order to apply the diaphragm in a structure model.

Diaphragm settings

The diaphragm is completely defined by means of:

- basic geometrical parameters,
- settings determining its position in a model.

The settings for the diaphragm are:

Name	Name of diaphragm settings set.
Diaphragm Lib	Type of defined diaphragm.
k	The value of coefficient k depends on the number of diaphragms: k = 2 for 1 or 2 lateral diaphragms, k = 4 for 3 or more diaphragms.
Diaphragm position	The position of the diaphragm may be either positive or negative. **Positive** means that the diaphragm is assembled in a way so that the width is greater at the top side. **Negative** means that the diaphragm is assembled in a way so that the width is greater at the bottom side.
Bolt position	Bolts may be located either at the top or bottom side of the diaphragm.
Bold pitch	Bolts may be either: in every rib (i.e. "br"), in each second rib (i.e. "2 br").
Frame distance	The distance of frames (i.e. the span of transverse bonds).
Length	The length of the transverse bond.
Position x1	Value x1 specifies the begin-point of the diaphragm on the beam.
Position x2	Value x1 specifies the end-point of the diaphragm on the beam.
Co-ordinate definition	Defines the co-ordinate system in which the **position x** is inputted.
Origin	Defines the origin from which the **position x** is measured.

Note: This set of parameters is not available for checking to AISC-ASD, Korean Standard and GBJ 17-88.

Note: For more info, see chapter Use of diaphragms in Steel Code Check Theoretical Background.

Links

Link settings

For built-up members, parameters defining links connecting individual partial cross-section of the member must be defined.

Division	Specifies the number of links per member.
Distance from begin	Defines the distance of the first link from the starting point of the beam.
Distance from end	Defines the distance of the last link from the end point of the beam.
Width of links	Defines the width of the links.
Thickness	Defines the thickness of the links.
On begin	Tells whether the first link is applied.
On end	Tells whether the very last link is applied.

Note: This set of parameters is available for checking to EC3, NEN6770/6771, DIN 18800 and Czech standard. For more info, see chapter **Battened compression member** of the relevant code in Steel Code Check Theoretical Background.

Adjusting the parameters for code checks

Adjusting the basic parameters

The procedure for adjustment of basic parameters

1. Open service **Steel**:
    a. either using tree menu function **Steel**,
    b. or using menu function **Tree > Steel**.
2. Select function **Setup** and open it.
3. In the dialogue that appears on the screen, select tab **Setup for check of steel members**.
4. Type required values and select appropriate options.
5. Confirm with **[OK]**.

Adjusting the default buckling parameters

The program provides for pre-setting of buckling parameters. These parameters are considered as default values whenever a new members is being added into the project.

The procedure for adjustment of default buckling parameters

1. Open service **Steel**:
    a. either using tree menu function **Steel**,
    b. or using menu function **Tree > Steel**.
2. Select function **Setup** and open it.
3. In the dialogue that appears on the screen, select tab **Setup of buckling**.
4. Type required values and select appropriate options.
5. Confirm with **[OK]**.

Note: If necessary, the parameters of particular members may be set different from the default.

Adjusting the buckling parameters for particular members

When a new member is being defined it takes the current default settings for buckling parameters. If necessary, it is possible to adjust unique parameters for each particular member.

The procedure for adjustment of buckling parameters for a particular member

1. In the graphical window, select the beam (or beams) whose buckling settings should be modified.
2. The beam properties are displayed in the Property window.
3. In the table cell **System lengths** use the combo box to select the required **System length definition** and go to the last step of the procedure.
4. If the required **System length definition** has not been defined yet, use the button at the right hand side of the cell to create a new **System length definition**.
5. Press button **Edit buckling** to open the editing dialogue.
6. Adjust required parameters.
7. Confirm with **[OK]**.
8. Clear the selection.

Note: See also Buckling parameters related to a particular beam.

Adjusting the deflection parameters

The procedure for adjustment of deflection parameters

1. Open service **Steel**:
    a. either using tree menu function **Steel**,
    b. or using menu function **Tree > Steel**.
2. Select function **Setup** and open it.
3. In the dialogue that appears on the screen, select tab **Setup of relative deformation**.
4. Type required values and select appropriate options.
5. Confirm with **[OK]**.

Adjusting the general fire resistance parameters

The procedure for adjustment of fire resistance parameters

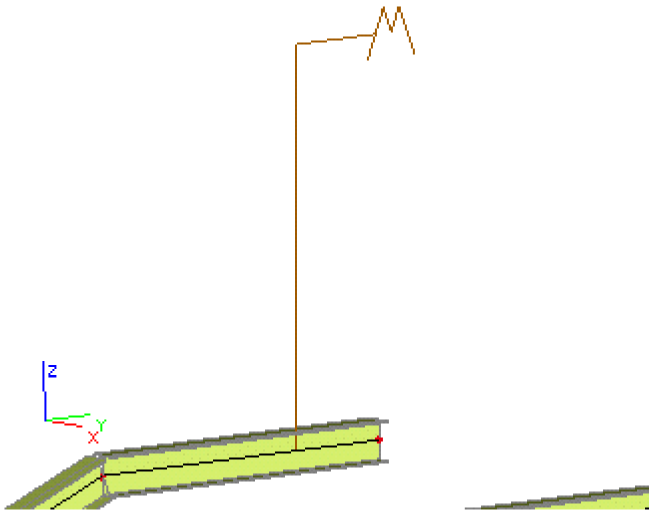
1. Open service **Steel**:
    a. either using tree menu function **Steel**,
    b. or using menu function **Tree > Steel**.
2. Select function **Setup** and open it.
3. In the dialogue that appears on the screen, select tab **Setup of fire resistance**.
4. Type required values and select appropriate options.
5. Confirm with **[OK]**.

Adjusting the fire resistance parameters for a member

For each member, the user mat specify concrete parameters related to the fire resistance of the particular member.

The procedure for adjustment of member data

1. Open service **Steel**:
    a. either using tree menu function **Steel**,
    b. or using menu function **Tree > Steel**.
2. Select function **Fire resistance** and open it.
3. Type required values and select appropriate options in.
4. Confirm with **[OK]**.
5. Select beams to which the adjusted values should be applied.
6. Close the function.
7. A special mark is attached to the selected beams telling that member data have been defined on them. The marks (member data) remain selected.
8. If required, clear the selection.



Defining a new insulation type

The procedure for definition of a new insulation type

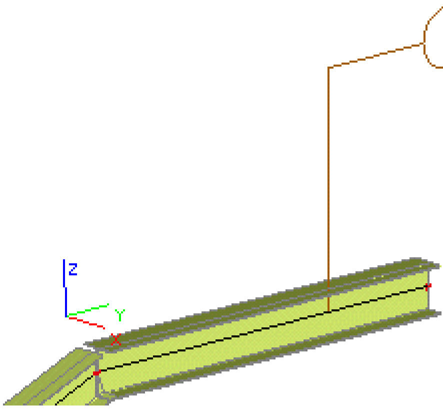
1. Open the database manager for **Insulations**:
   a. either using tree menu function **Library > Insulations**,
   b. or using menu function **Libraries > Insulations**.
2. Click button **[New]** to create a new insulation type.
3. A new insulation type is added into the list of defined insulations.
4. Click button **[Edit]** to edit its properties.
5. Input the parameters.
6. Confirm with **[OK]**.
7. Close the database manager.

Note: A new insulation type is defined in the program tool called Database manager. The database manager can be used not only for the definition of a new insulation type, but also for editing of existing ones, for removal of no more necessary ones, and for other operations related to the management of insulation database.

Adjusting the member settings

The procedure for adjustment of member data

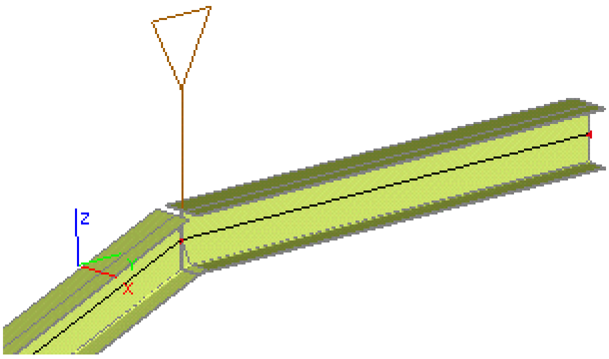
1. Open service **Steel**:
   a. either using tree menu function **Steel**,
   b. or using menu function **Tree > Steel**.
2. Select function **Member data** and open it.
3. Type required values and select appropriate options in.
4. Confirm with **[OK]**.
5. Select beams to which the adjusted values should be applied.
6. Close the function.
7. A special mark is attached to the selected beams telling that member data have been defined on them. The marks (member data) remain selected.
8. If required, clear the selection.



Adjusting the lateral torsional buckling settings

The procedure for definition of lateral torsional restraints

1. Open service **Steel**:
    a. either using tree menu function **Steel**,
    b. or using menu function **Tree > Steel**.
2. Select function **LTB Restrain** and open it.
3. Type required values and select appropriate options in.
4. Confirm with **[OK]**.
5. Select beams to which the adjusted values should be applied.
6. Close the function.
7. A special mark is attached to the selected beams telling that LTB restraints have been defined on them. The marks remain selected.
8. If required, clear the selection.

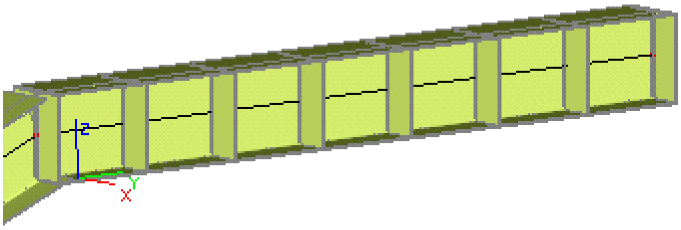


Adjusting the stiffener settings

The procedure for definition of diaphragm settings

1. Open service **Steel**:
    a. either using tree menu function **Steel**,
    b. or using menu function **Tree > Steel**.
2. Select function **Stiffeners** and open it.
3. Type required values and select appropriate options in.
4. Confirm with **[OK]**.

5. Select beams to which the adjusted values should be applied.
6. Close the function.
7. The stiffeners are displayed along the beams. The displayed stiffeners remain selected.
8. If required, clear the selection.



Defining a new diaphragm

The procedure for definition of a new diaphragm

1. Open the database manager for **Diaphragms**:
    a. either using tree menu function **Library > Diaphragms**,
    b. or using menu function **Libraries > Diaphragms**.
2. Click button **[New]** to create a new diaphragm.
3. A new diaphragm is added into the list of defined diaphragms.
4. Click button **[Edit]** to edit its properties.
5. Input the parameters.
6. Confirm with **[OK]**.
7. Close the database manager.

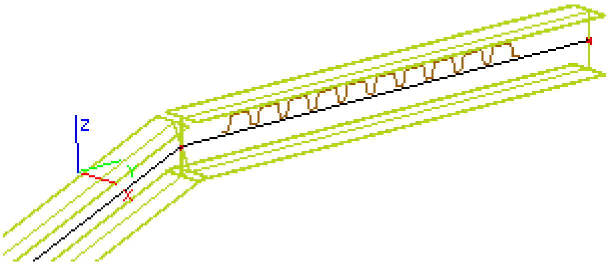
Note: A new diaphragm is defined in the program tool called Database manager. The database manager can be used not only for the definition of a new diaphragm, but also for editing of existing ones, for removal of no more necessary ones, and for other operations related to the management of diaphragm database.

Adjusting the diaphragm settings

The procedure for definition of diaphragm settings

1. Open service **Steel**:
    a. either using tree menu function **Steel**,
    b. or using menu function **Tree > Steel**.
2. Select function **Diaphragms** and open it.
3. Type required values and select appropriate options in.
4. Confirm with **[OK]**.
5. Select beams to which the adjusted values should be applied.
6. Close the function.
7. A special mark is attached to the selected beams telling that diaphragms have been defined on them. The marks remain selected.

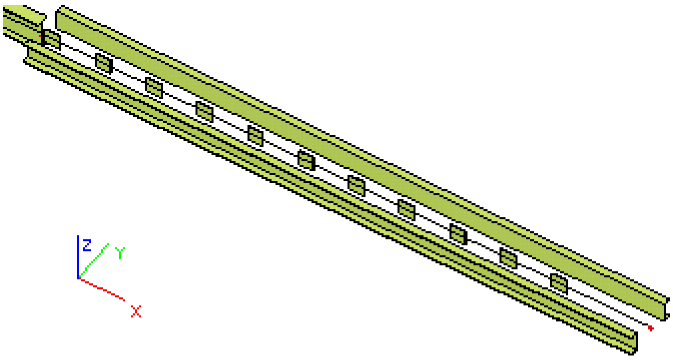
8. If required, clear the selection.



Adjusting the link settings

The procedure for definition of diaphragm settings

1. Open service **Steel**:
   a. either using tree menu function **Steel**,
   b. or using menu function **Tree > Steel**.
2. Select function **Links** and open it.
3. Type required values and select appropriate options in.
4. Confirm with **[OK]**.
5. Select beams to which the adjusted values should be applied.
6. Close the function.
7. The links are displayed along the beams. The displayed links remain selected.
8. If required, clear the selection.



Note: This set of parameters is available for checking to EC3, NEN6770/6771, DIN 18800 and Czech standard.

Performing the checks

Prerequisites of the check

Before the user may get down to the business of checking, a set of specific conditions must be met.

1. The model of the analysed structure must be properly defined.
2. The boundary conditions and loads reflecting the real conditions of the structure must be specified.
3. The model of the analysed structure must be calculated, in other words, the internal forces and deformations must be known.

General principles of checking

The procedure used for performing the checking is analogous to the procedure for evaluation of results.

It can be summarised by the following points:

1. Opening of the required service.
2. Selection of beams that should be checked.
3. Selection of load case or load case combination that should be used.
4. Adjusting of display parameters.
5. Selection of values to be displayed.
6. Displaying of the results of the checking.

Service Check

The required type of check can be selected in the tree menu of service **Steel > Beams**. Once the type of the check is selected, the appropriate parameters are listed in the Property window.

Parameters common for most of the available check types are:

Selection	The user may display the results either on all or only selected beams.
Load type	Specifies what "load type" is considered for the display. Available load types are: load cases, load case combinations, result classes.
Load case / combination / class	For each of the above specified load type a set of available items (load cases, combinations, result classes) is offered.
Filter	The set of beams where the results are displayed may be specified by means of a filter.
Values	For each of the result groups (unity check, fire resistance check, etc.) a set of quantities is offered for display. The user may select which one is really shown.
Extreme	The numerical values may be displayed in specified extreme points.
Drawing setup	It is possible to adjust the style of the diagrams.
Other specific parameters	Some of the available result groups (unity check, fire resistance check, etc.) may have other group-specific parameters.

Selection of members

The result diagrams may be displayed on:

- all the beams in the structure,
- selected beams only.

Which variant is actually applied can be adjusted in the Property window by means of parameters **Selection** and **Filter**.

Selection

All	If this option is selected, the result diagrams are displayed on all beams in the structure.
User	If this option is chosen, the user must make a selection to define the beams for the display of result diagrams.

The selection must be ended (using **[Esc]** key or pop-up menu function **End of command**) before it may be applied.

Filter

No	No filter is applied.
Wildcard	The set of beams for display is defined by a wildcard expression. E.g. expression "N*" lists all entities whose name starts with letter N. The expression "B??" lists all entities whose name starts with letter B and is followed by two characters.
Cross-section	Diagrams are shown only on entities of selected cross-section.
Material	Diagrams are shown only on entities of selected material.
Layer	Diagrams are shown only on entities inserted into selected layer.

Displaying the results after re-adjustment of check parameters

Whenever the settings in the Property window of service Check are changed, the diagrams on the screen usually require regeneration. Because the fully automatic regeneration could be very slow for excessive models, it is up to the user to regenerate the drawing when necessary.

Any time the user makes a change that affects the display, the program paints the cell **Redraw** of the Property window in red colour. Until the user presses the button **[Redraw]**, the cell remains highlighted.

Performing the resistance check

The resistance check offers the user to select from the following variants:

- unity check,
- section check,
- stability check.

The procedure for performing the check

1. Open service **Steel**:
    a. either using tree menu function **Steel**,
    b. or using menu function **Tree > Steel**.
2. Select function **Check**.
3. In the **Property window** select the values that should be displayed and adjust the other parameters as required.
4. The diagrams are displayed on the screen.

Note 1: More information about displaying of results can be found in chapter Results > Displaying the internal forces in the Reference manual for SCIA.ESA PT.

Note 2: If a detailed checking of a single beam is required, the user may apply the procedure for a separate checking of individual beams described later in this book.

Performing the slenderness check

Values for display

When performing the slenderness check, the user may evaluate the following values:

Ly	System length for buckling around y axis

ky	Buckling ratio (used during code check) for buckling around y axis
ly	Buckling length for buckling around y axis ly = Ly * ky
Lam y	Slenderness around y axis $\lambda_y = \frac{l_y}{i}$ ly : buckling length for buckling around y axis iy : radius of gyration around z axis
Lz	System length for buckling around z axis
kz	Buckling ratio (used during code check) for buckling around z axis
lz	Buckling length for buckling around z axis lz = Lz * kz
Lam z	Slenderness around z axis $\lambda_z = \frac{l_z}{i}$ lz : buckling length for buckling around z axis iz : radius of gyration around z axis
l LTB	The lateral-torsional buckling length l LTB = kLTB * L LTB

Second order buckling ratio

Second order buckling ratio of the slenderness check can be obtained by means of:

- linear calculation,
- second order calculation.

The procedure

The procedure for performing the slenderness check

1. Open service **Steel**:
   a. either using tree menu function **Steel**,
   b. or using menu function **Tree > Steel**.
2. Select function **Slenderness**.
3. In the **Property window** select the values that should be displayed and adjust the other parameters as required.
4. The diagrams are displayed on the screen.

Note: More information about displaying of results can be found in chapter Results > Displaying the internal forces in the Reference manual for SCIA.ESA PT.

Performing the fire resistance check

The procedure for performing the fire resistance check

1. Open service **Steel**:
   a. either using tree menu function **Steel**,
   b. or using menu function **Tree > Steel**.
2. Select function **Check – Fire resistance**.
3. In the **Property window** select the values that should be displayed and adjust the other parameters as required.
4. The diagrams are displayed on the screen.

Note: More information about displaying of results can be found in chapter Results > Displaying the internal forces in the Reference manual for SCIA.ESA PT.

Performing the relative deformation check

Sections

The user may select the sections where the check is performed.

All	The diagram of relative deformation is drawn in all sections along the evaluated beams.
End	The diagram of relative deformation is drawn only in all end-sections of the evaluated beams.

The procedure for performing the relative deformation check

1. Open service **Steel**:
    a. either using tree menu function **Steel**,
    b. or using menu function **Tree > Steel**.
2. Select function **Relative deformation**.
3. In the **Property window** select the values that should be displayed and adjust the other parameters as required.
4. The diagrams are displayed on the screen.

Note: More information about displaying of results can be found in chapter Results > Displaying the internal forces in the Reference manual for SCIA.ESA PT.

Displaying the results in tabular form

Preview of check results

The results of any check may be displayed in the form of readable tables in the Preview window.

The procedure for the insertion of a table with check results into the Preview window

1. Perform the required type of check.
2. In the property window, select the required level of output:
    a. brief,
    b. normal,
    c. detailed.
3. 3. Call function **Print / Preview table**:
    a. using menu function **File > Print data > Print / Preview table**,
    b. using function **Print data > Print / Preview table** on toolbar **Project**
4. 4. The results are displayed in the **Preview window**.

Check results in the Document

The results of any check may be displayed in the Document in the form of readable tables. Later the document may be edited in a way so that the final report looks as required by those who receive it..

The procedure for the insertion of a table with check results into the Document

1. Perform the required type of check.
2. In the property window, select the required level of output:
    a. brief,
    b. normal,
    c. output.
3. Call function **Table to document**:
    a. using menu function **File > Print data > Table to document**,

b. using function **Print data > Table to document** on toolbar **Project**

4. The results are inserted into the **Document**.

Checking of a single beam

When performing one of below listed checks, the user may prefer to see the detailed results of the check for one member at a time.

This option is available if item **Single check** is displayed in the **Property window** after the check function has been activated.

The option is available for:

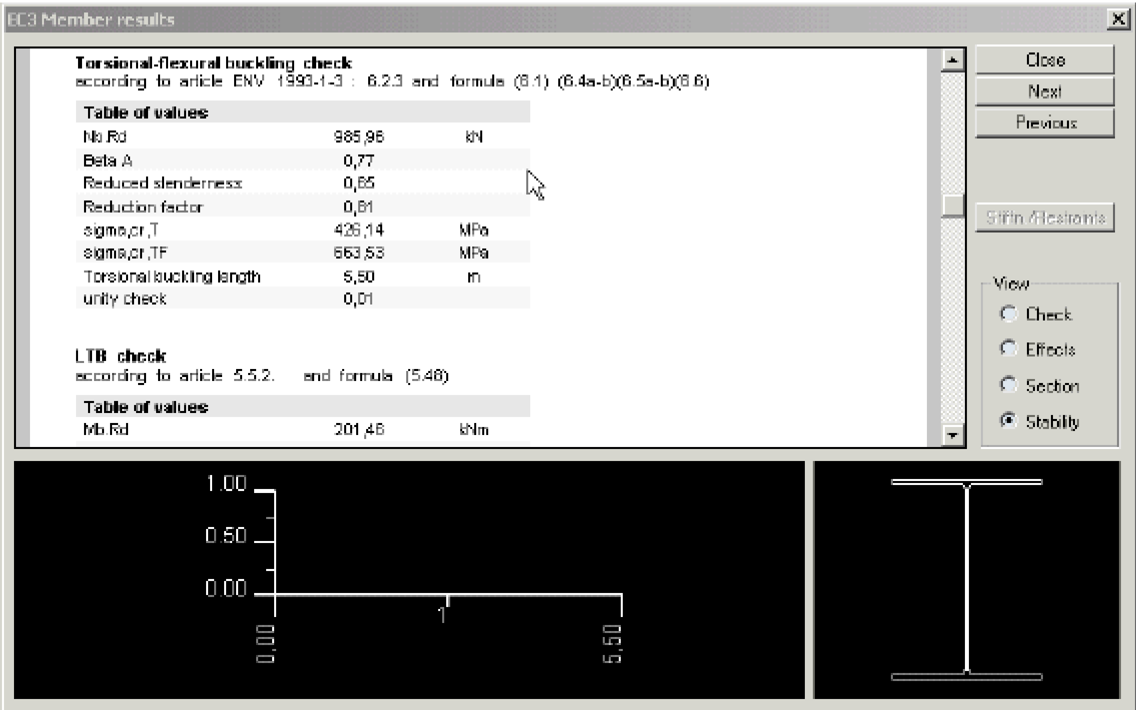
- unity check,
- section check,
- stability check.

For information about these checks see chapter Performing the resistance check.

Results of checking for a single beam

After the button under item **Single check** is pressed a new dialogue window is opened on the screen.

The window may look like:



Parts of the "single-check" window

report window	This part of the dialogue displays all the results available for the selected check.
longitudinal section	This part may display results in the form of diagram.
cross-section	Here, the cross-section of the checked beam is shown.
control buttons	The control buttons provide access to various information.

Control buttons

Close	It closes the "single-check" dialogue.
Next	It displays results of check of next beam in the structure.
Previous	It displays results of check of previous beam in the structure.
Check	It displays in the report window the overall check results.

Effect	It displays in the report window load which the beam is subject to.
Section	It displays in the report window results of section check for the selected beam.
Stability	It displays in the report window results of stability check.

Optimisation

Introduction to optimisation

Once a structure has been designed and calculated, it is the time to perform checking and usually a kind of optimisation of the original design.

SCIA.ESA PT contains a powerful tool for this task. The optimisation of applied profiles may be done automatically or semi-automatically. The process of optimisation results in what may be called an economical and good solution.

The optimisation process in SCIA.ESA PT is based on assumptions given in the following chapter.

Principles of optimisation

An optimisation in general represents a complex task. A full, complete and really "optimal" optimisation would usually lead to a long and often recursive process. Therefore, SCIA.ESA PT implements a kind of compromise.

One optimisation step takes account of a single cross-section only

It is possible to optimise one cross-section at a time. The user selects the cross-section from a list of all cross-sections applied in the structure.

One optimisation step considers only "selected" members

It is possible to limit the optimisation process to only a selected set of members. The user may make a selection to specify which beams of the given cross-section should be considered for the optimisation calculations.

One optimisation step affects the whole structure

Once the optimised cross-section is found, it is applied to ALL members in the structure that are of the specified cross-section. It is of no importance whether the optimisation calculation was limited to a selected number of beams or not. The final effect of the optimisation is that the original cross-section is simply replaced with the new, i.e. optimised, cross-section.

Optimisation parameters for rolled cross-sections

The user may control the process of optimisation by means of a set of parameters.

Check parameter

Maximal check	This parameter tells the program what is the maximal allowable value for satisfactory checking.
Maximum unity check	This item shows the found maximal check result for the optimised cross-section.

Shape parameters for optimisation

Sort by height	The sequence of cross-section is based on the height.
Sort by A (sectional area)	The sequence of cross-section is based on the sectional area.
Sort by Iy (moment of inertia)	The sequence of cross-section is based on the moment of inertia.

Buttons for manual optimisation

Set value	This button enables the user to set manually the required value of selected dimension (see above).

Next down	This button finds one-step smaller cross-section according to defined shape parameters (see above).
Next up	This button finds one-step larger cross-section according to defined shape parameters (see above).

Buttons for automatic optimisation

Search for optimal	This button finds automatically the optimal cross-section.

Optimisation parameters for welded and solid cross-sections

The user may control the process of optimisation by means of a set of parameters.

Check parameter

Maximal check	This parameter tells the program what is the maximal allowable value for satisfactory checking.
Maximum unity check	This item shows the found maximal check result for the optimised cross-section.

Shape parameters for optimisation

Dimension	This item determines which of the dimensions of the cross-section should be optimised. All other dimensions remain unchanged.
Step	This item specifies the step by which the selected dimension id modified in order to give one-step smaller or larger cross-section.
Minimum	This item specifies the minimal size of the selected dimension.
Maximum	This item specifies the maximal size of the selected dimension.

Buttons for manual optimisation

Set value	This button enables the user to set manually the required value of selected dimension (see above).
Next down	This button finds one-step smaller cross-section according to defined shape parameters (see above).
Next up	This button finds one-step larger cross-section according to defined shape parameters (see above).

Buttons for automatic optimisation

Search for optimal	This button finds automatically the optimal cross-section.

Optimising the members

It is possible to perform both automatic and manual optimisation. The process for both is identical except the last but one step. Therefore, only one procedure will be given here in detail. The other one will be explained briefly.

The procedure for the automatic optimisation of members

1. Open service **Steel**.
2. Open branch **Beams**.
3. Select function **Check**.
4. In the Property window, go to item **Filter** and set it to **Cross-section**.
5. In the Property window, go to item **Cross-section** and select the one you want to optimise.

6. In the Property window, go to item **Selection** and set it to **User** or **All**, depending on your requirements.
7. If the item is set to **User**, make the selection and press button **[Esc]** to close the selection.
8. If the item **Selection** has been re-adjusted, press button **[Redraw]** in order to refresh the screen and see the appropriate display.
9. In the Property window, go to item **Optimisation** and press the button there.
10. The optimisation dialogue is opened on the screen.
11. Adjust the parameters (of rolled or welded profile) as required.
12. Press button **[Search for optimal]**. The program finds the optimal cross-section.
13. If you agree, press **[OK]** to confirm.

The procedure for the manual optimisation of members

The procedure is identical except step 12.

In manual optimisation, the user must press (repeatedly, if required) buttons **[Next down]** and **[Next Up]**, in order to find the optimal cross-section. Alternatively, it is also possible to set the required value directly by means of button **[Set value]**.

Note: The project must be calculated beforehand.

Theoretical background

Theoretical background

The theoretical background is given in a separate volume Steel Check Code Theoretical Background.