Robot Structural Analysis Professional

AUTODESK

VERIFICATION MANUAL

FOR EUROCODES

INTRODUCTION

This verification manual contains numerical examples for structures prepared and originally calculated by **Autodesk Robot Structural Analysis Professional version 2023**. The comparison of results is still valid for the next versions.

All examples have been taken from handbooks that include benchmark tests covering fundamental types of behaviour encountered in structural analysis. Benchmark results (signed as “Handbook”) are recalled and compared with results of Autodesk Robot Structural Analysis Professional (signed further as “Robot”).

Each example contains the following parts:

- title of the problem
- specification of the problem
- Robot solution to the problem
- outputs with calculation results and calculation notes
- comparison between Robot results and exact solution
- conclusions.

STEEL

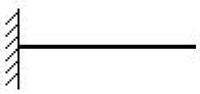
1. Eurocode 3 (EN 1993-1-1:2005)

VERIFICATION EXAMPLE 1
- Axial compression

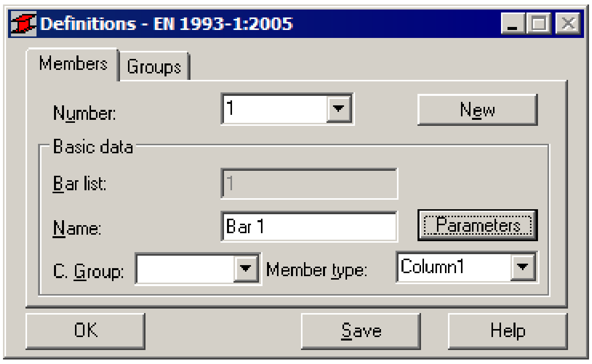
Example taken from Designer's Guide to EN 1993-1-1
L.Gardner and D.A.Nethercot, Thomas Telford Publishing, 2005

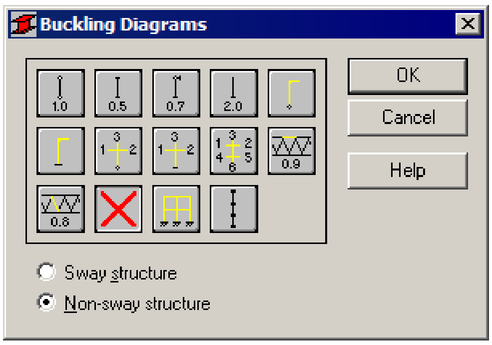
TITLE:
Axial compression (Example 6.2 page 44).

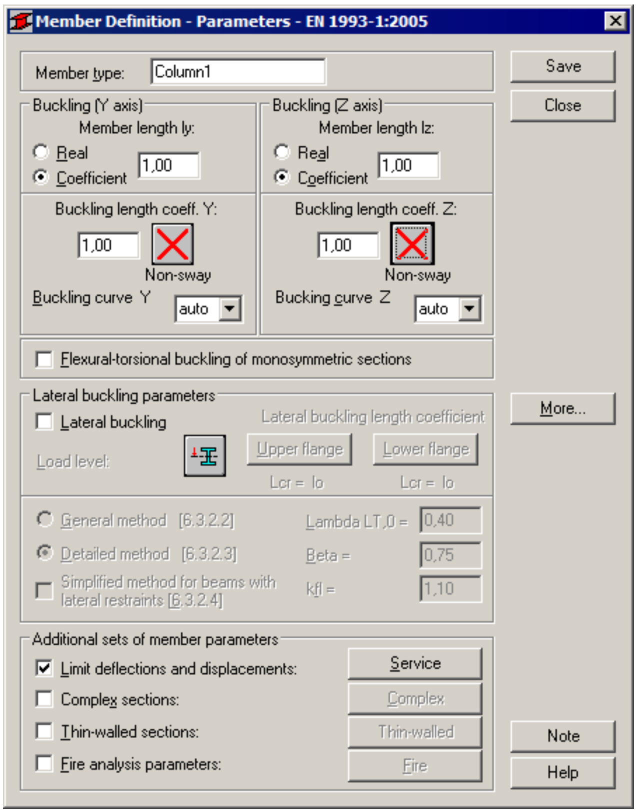
SPECIFICATION:
The member shown below is a cantilever. The design compression resistance force N_{Sd} = 3305 kN is checked for the assumed section UC 254x254x73, steel grade S355.



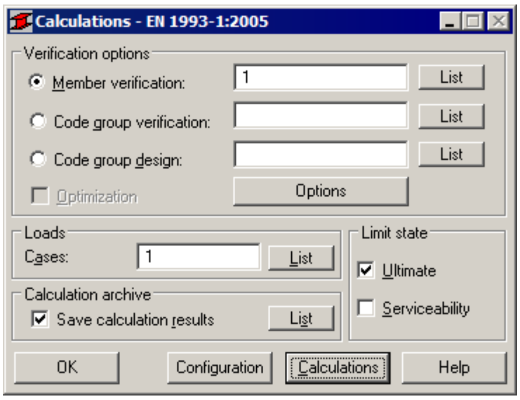
SOLUTION:
Define a new type of member. For analyzed member pre-defined type of member COLUMN may be initially opened. Press the *Parameters* button in DEFINITIONS/MEMBERS tab, which opens MEMBER DEFINITION – PARAMETERS dialog. Type a new name **Column 1** in the *Member Type* editable field. Then, press *Buckling Length coefficient Y* icon and select the twelfth icon (*no buckling*). For *Z* direction press *Buckling Length coefficient Z* and choose the same icon. Save the newly created type of member.



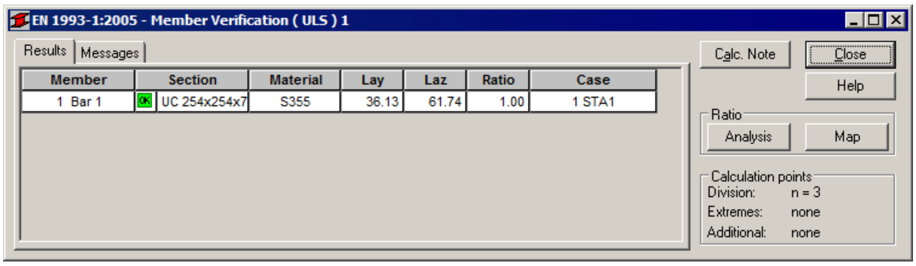




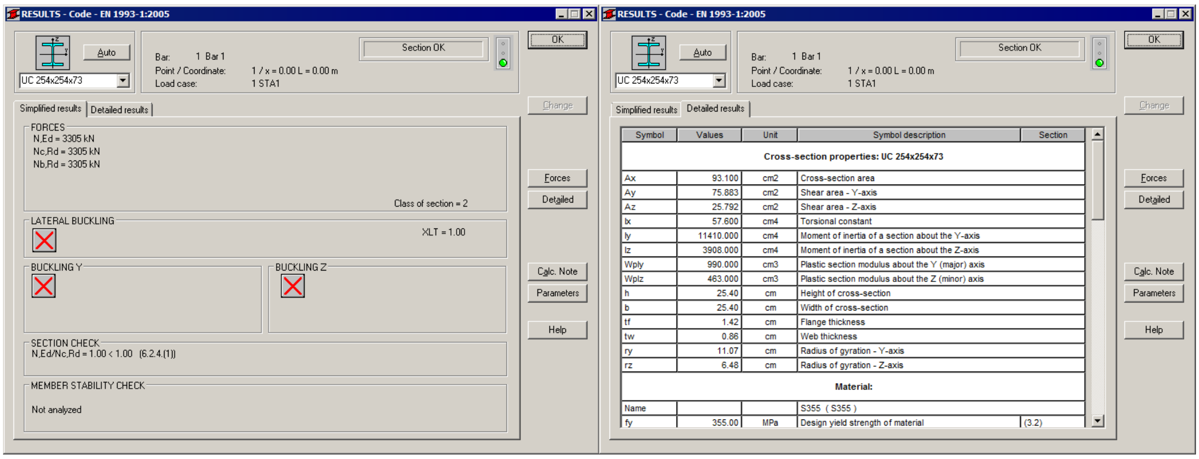
In the CALCULATIONS dialog set *Member Verification* option for member 1 and switch off *Limit State – Serviceability* (only Ultimate Limit state will be analyzed). Now, start the calculations by pressing *Calculations* button.



Member Verification dialog with most significant results data will appear on screen. Pressing the line with results for member 1 opens the RESULTS dialog with detailed results for the analyzed member.



The view of the RESULTS dialog is presented below. Moreover, the printout note containing the same results data as in *Simplified results* tab of the RESULTS dialog is added.



AUTODESK

STEEL DESIGN

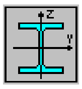
CODE: *EN 1993-1:2005, Eurocode 3: Design of steel structures.*
ANALYSIS TYPE: Member Verification

CODE GROUP:
MEMBER: 1 Bar 1 **POINT:** 1 **COORDINATE:** x = 0.00 L = 0.00 m

---

LOADS:
Governing Load Case: 1 STA1

MATERIAL:
S355 (S355) fy = 355.00 MPa



SECTION PARAMETERS: UC 254x254x73

h=25.40 cm	gM0=1.00	gM1=1.00	
b=25.40 cm	Ay=75.883 cm2	Az=25.792 cm2	Ax=93.100 cm2
tw=0.86 cm	Iy=11410.000 cm4	Iz=3908.000 cm4	Ix=57.600 cm4
tf=1.42 cm	Wply=990.000 cm3	Wplz=463.000 cm3	

INTERNAL FORCES AND CAPACITIES:
N,Ed = 3305 kN
Nc,Rd = 3305 kN
Nb,Rd = 3305 kN

Class of section = 2

LATERAL BUCKLING PARAMETERS:

BUCKLING PARAMETERS:

About Y axis: About Z axis:

VERIFICATION FORMULAS:
Section strength check:
N,Ed/Nc,Rd = 1.00 < 1.00 (6.2.4.(1))

Section OK !!!

COMPARISON:

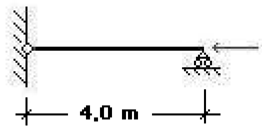
Resistance, interaction expression	Robot	Handbook
1. design compression resistance of the cross-section $N_{c.Rd}$	3305	3305

VERIFICATION EXAMPLE 2
- Axial compression with buckling

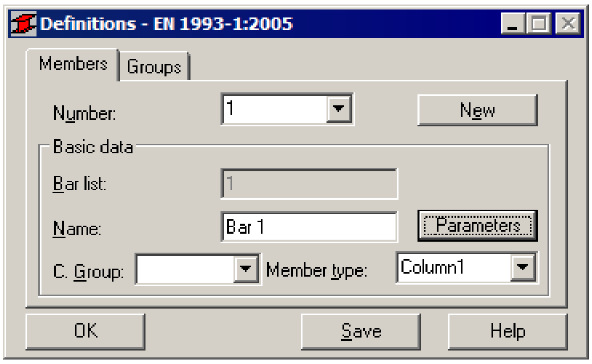
Example taken from Designer's Guide to EN 1993-1-1
L.Gardner and D.A.Nethercot, Thomas Telford Publishing, 2005

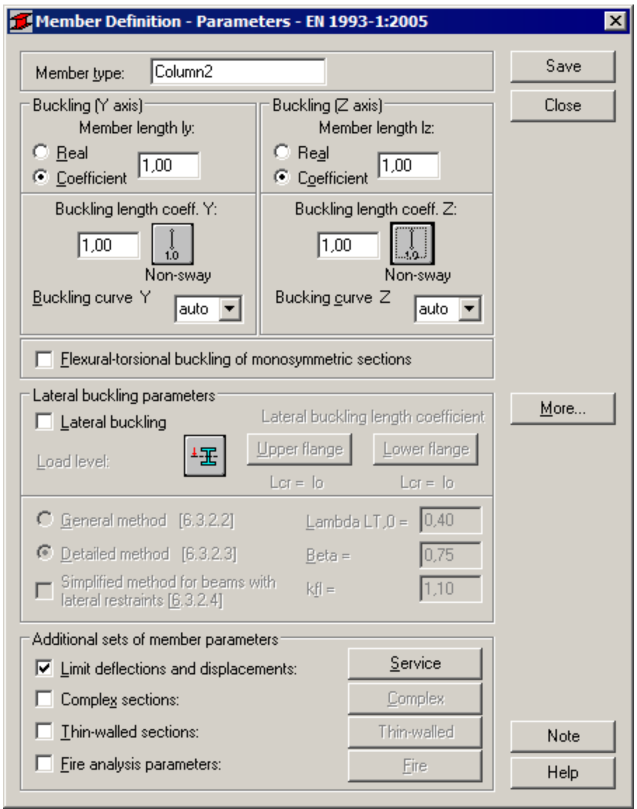
TITLE:
Buckling resistance of a compression member (Example 6.7 page 66).

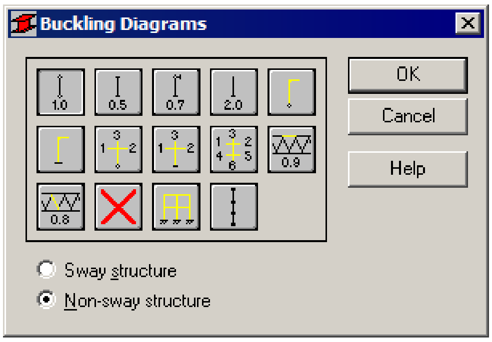
SPECIFICATION:
The member shown below has pinned boundary conditions. The design compression force N = 1630 kN is checked for the assumed circular hollow section CHS 244,5x10, steel grade S275.



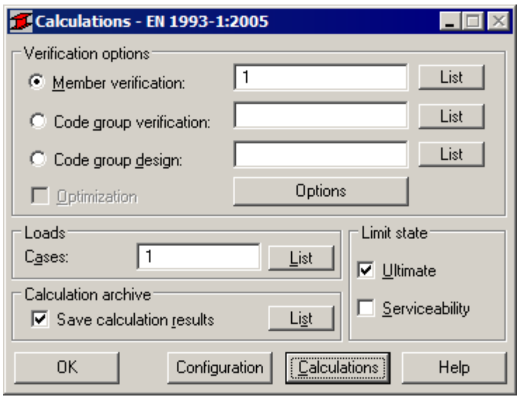
SOLUTION:
Define a new type of member. For analyzed member pre-defined type of member COLUMN may be initially opened. Press the *Parameters* button in DEFINITIONS/MEMBERS tab, which opens MEMBER DEFINITION – PARAMETERS dialog. Type a new name **Column 2** in the *Member Type* editable field. The *Buckling Length coefficient Y* and *Z* are set to the buckling length *1.0*. Save the newly created type of member.



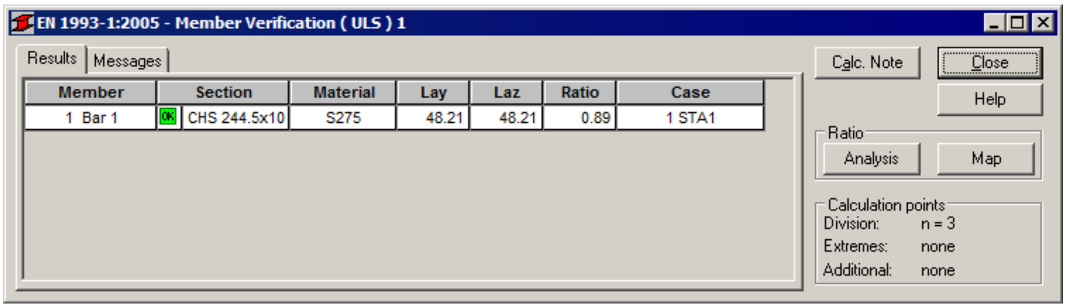




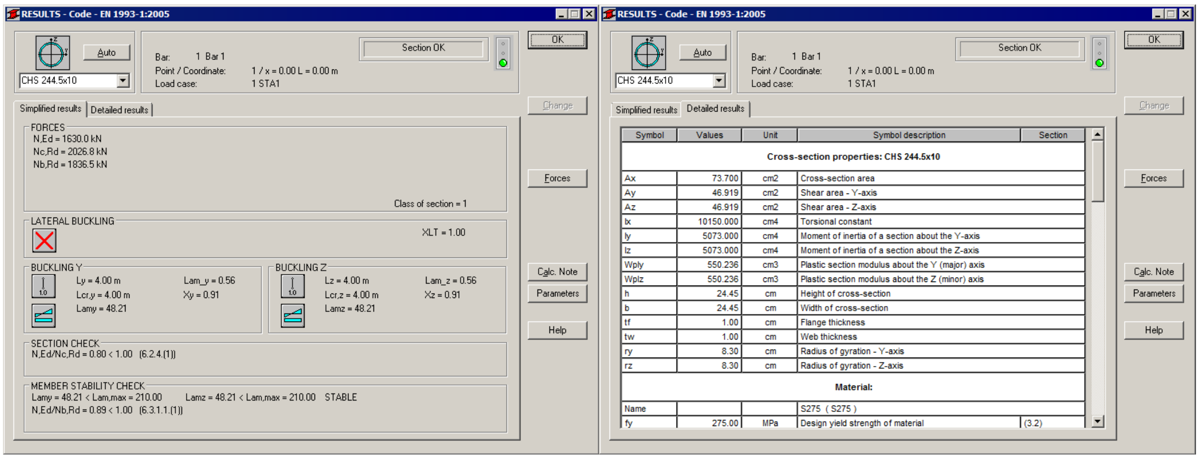
In the CALCULATIONS dialog set *Member Verification* option for member 1 and switch off *Limit State – Serviceability* (only Ultimate Limit state will be analyzed). Now, start the calculations by pressing *Calculations* button.



Member Verification dialog with most significant results data will appear on screen. Pressing the line with results for member 1 opens the RESULTS dialog with detailed results for the analyzed member.



The view of the RESULTS dialog is presented below. Moreover, the printout note containing the same results data as in *Simplified results* tab of the RESULTS dialog is added.



STEEL DESIGN

CODE: *EN 1993-1:2005, Eurocode 3: Design of steel structures.*
ANALYSIS TYPE: Member Verification

CODE GROUP:
MEMBER: 1 Bar 1 **POINT:** 1 **COORDINATE:** x = 0.00 L = 0.00 m

---

LOADS:
Governing Load Case: 1 STA1

MATERIAL:
S275 (S275) fy = 275.00 MPa

SECTION PARAMETERS: CHS 244.5x10

h=24.45 cm	gM0=1.00	gM1=1.00	
b=24.45 cm	Ay=46.919 cm2	Az=46.919 cm2	Ax=73.700 cm2
tw=1.00 cm	Iy=5073.000 cm4	Iz=5073.000 cm4	Ix=10150.000 cm4
tf=1.00 cm	Wply=550.236 cm3	Wplz=550.236 cm3	

INTERNAL FORCES AND CAPACITIES:
N,Ed = 1630.0 kN
Nc,Rd = 2026.8 kN
Nb,Rd = 1836.5 kN

Class of section = 1

LATERAL BUCKLING PARAMETERS:

BUCKLING PARAMETERS:

About Y axis:		About Z axis:	
Ly = 4.00 m	Lam_y = 0.56	Lz = 4.00 m	Lam_z = 0.56
Lcr,y = 4.00 m	Xy = 0.91	Lcr,z = 4.00 m	Xz = 0.91
Lamy = 48.21		Lamz = 48.21	

VERIFICATION FORMULAS:
Section strength check:
N,Ed/Nc,Rd = 0.80 < 1.00 (6.2.4.(1))
Global stability check of member:
Lambda,y = 48.21 < Lambda,max = 210.00 Lambda,z = 48.21 < Lambda,max = 210.00 STABLE
N,Ed/Nb,Rd = 0.89 < 1.00 (6.3.1.1.(1))

Section OK !!!

COMPARISON:

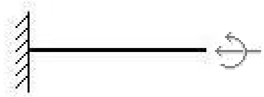
Resistance, interaction expression	Robot	Handbook
1. cross-section compression resistance $N_{c.Rd}$	2026.8	2026.8
2. non-demensional slenderness for flexural buckling Lambda	0,56	0,56

VERIFICATION EXAMPLE 3
- Combined compression and bending

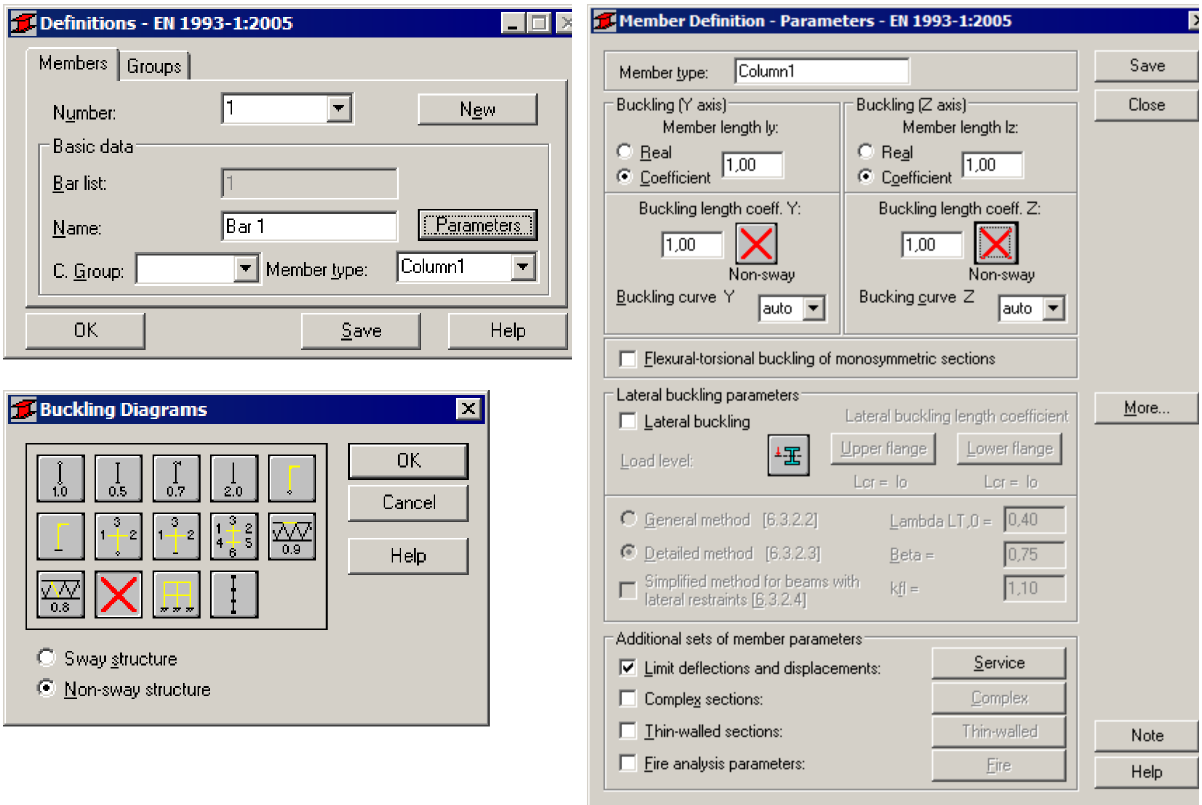
Example taken from Designer's Guide to EN 1993-1-1
L.Gardner and D.A.Nethercot, Thomas Telford Publishing, 2005

TITLE:
Combined compression and bending (Example 6.6 page 57).

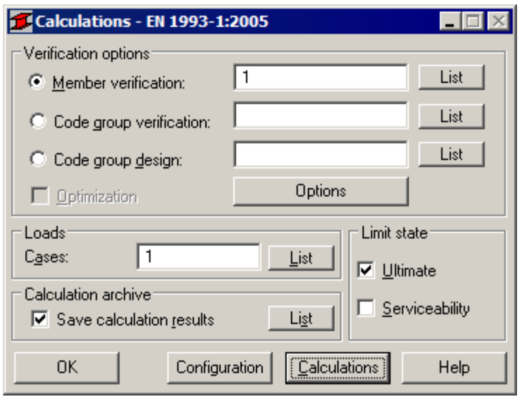
SPECIFICATION:
The member carry combined major axis bending moment and an axial force. The assumed section UB 457x191x98 in grade S235 steel is checked to determine the maximum bending moment in the presence of an axial force N = 1400 kN.



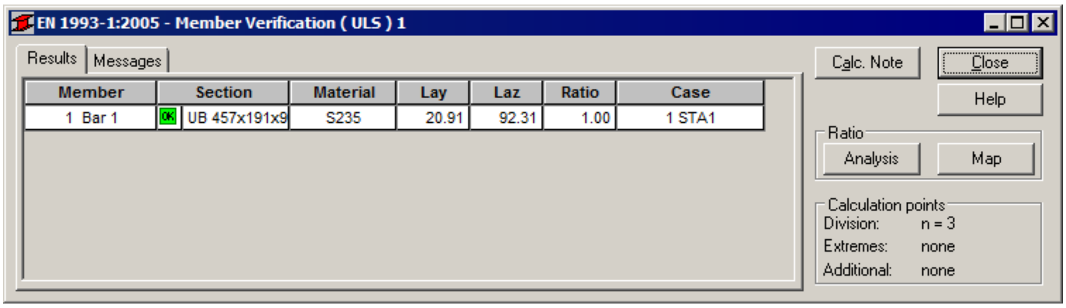
SOLUTION:
Define a new type of member. For analyzed member pre-defined type of member COLUMN may be initially opened. Press the *Parameters* button in DEFINITIONS/MEMBERS tab, which opens MEMBER DEFINITION – PARAMETERS dialog. Type a new name **Column 1** in the *Member Type* editable field. Then, press *Buckling Length coefficient Y* icon and select the twelfth icon (*no buckling*). For *Z* direction press *Buckling Length coefficient Z* and choose the same icon. Save the newly created type of member.



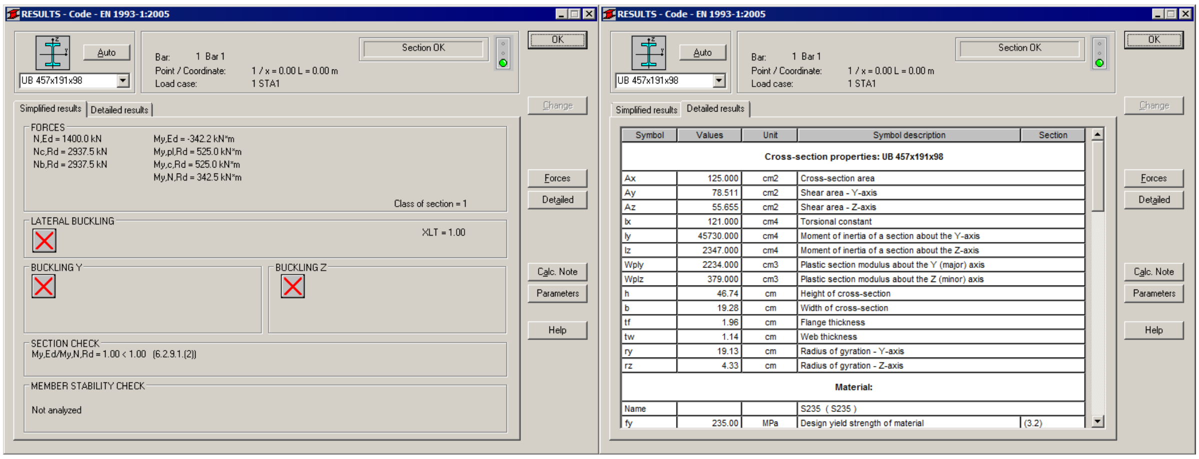
In the CALCULATIONS dialog set *Member Verification* option for member 1 and switch off *Limit State – Serviceability* (only Ultimate Limit state will be analyzed). Now, start the calculations by pressing *Calculations* button.



Member Verification dialog with most significant results data will appear on screen. Pressing the line with results for member 1 opens the RESULTS dialog with detailed results for the analyzed member.



The view of the RESULTS dialog is presented below. Moreover, the printout note containing the same results data as in *Simplified results* tab of the RESULTS dialog is added.



STEEL DESIGN

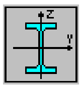
CODE: *EN 1993-1:2005, Eurocode 3: Design of steel structures.*
ANALYSIS TYPE: Member Verification

CODE GROUP:
MEMBER: 1 Bar 1 **POINT:** 1 **COORDINATE:** x = 0.00 L = 0.00 m

---

LOADS:
Governing Load Case: 1 STA1

MATERIAL:
S235 (S235) fy = 235.00 MPa



SECTION PARAMETERS: **UB 457x191x98**

h=46.74 cm	gM0=1.00	gM1=1.00	
b=19.28 cm	Ay=78.511 cm2	Az=55.655 cm2	Ax=125.000 cm2
tw=1.14 cm	Iy=45730.000 cm4	Iz=2347.000 cm4	Ix=121.000 cm4
tf=1.96 cm	Wply=2234.000 cm3	Wplz=379.000 cm3	

INTERNAL FORCES AND CAPACITIES:

N,Ed = 1400.0 kN	My,Ed = -342.2 kN*m
Nc,Rd = 2937.5 kN	My,pl,Rd = 525.0 kN*m
Nb,Rd = 2937.5 kN	My,c,Rd = 525.0 kN*m
	My,N,Rd = 342.5 kN*m

Class of section = 1

LATERAL BUCKLING PARAMETERS:

BUCKLING PARAMETERS:

About Y axis: About Z axis:

VERIFICATION FORMULAS:
Section strength check:
N,Ed/Nc,Rd = 0.48 < 1.00 (6.2.4.(1))
My,Ed/My,c,Rd = 0.65 < 1.00 (6.2.5.(1))
My,Ed/My,N,Rd = 1.00 < 1.00 (6.2.9.1.(2))

Section OK !!!

COMPARISON:

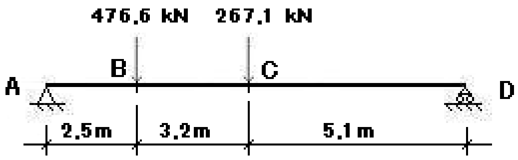
Resistance, interaction expression	Robot	Handbook
1. plastic moment resistance $M_{pl,y,.Rd}$	525,0	524,5
2. reduced plastic moment resistance $M_{N,y,.Rd}$	342,5	342,2

VERIFICATION EXAMPLE 4
- Bending with lateral buckling

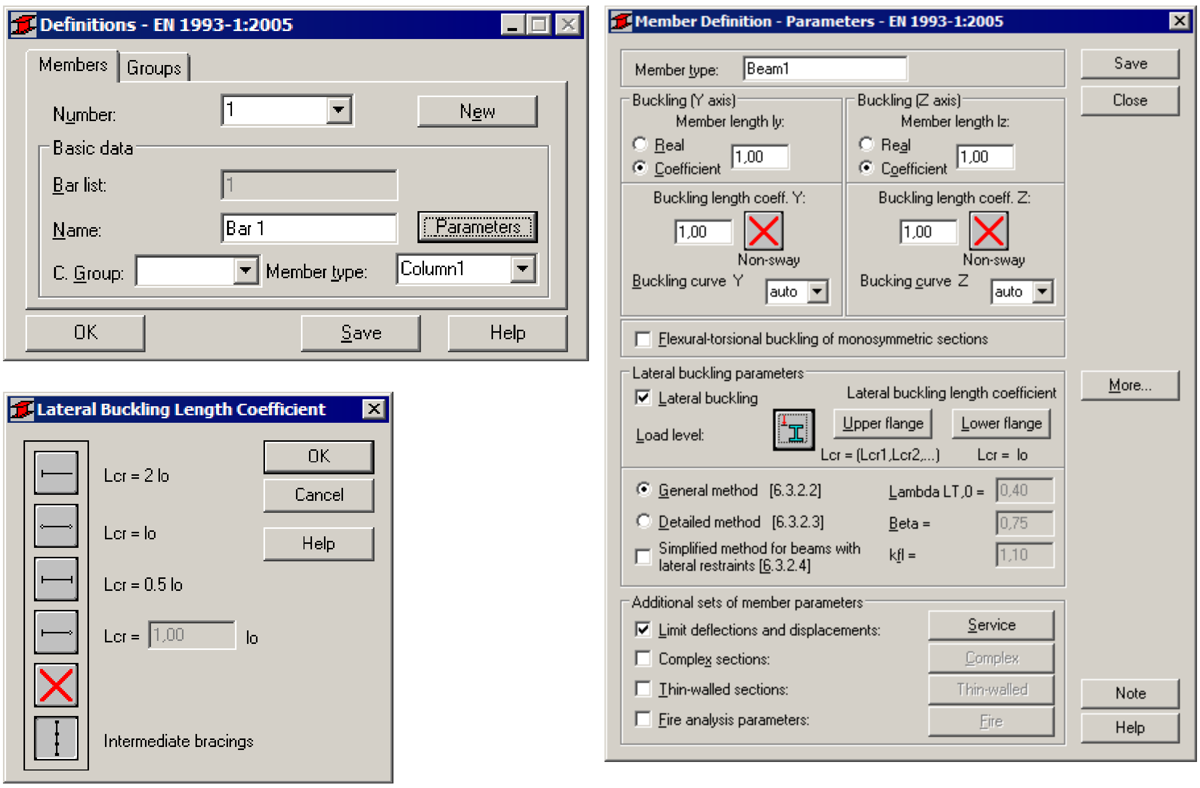
Example taken from Designer's Guide to EN 1993-1-1
L.Gardner and D.A.Nethercot, Thomas Telford Publishing, 2005

TITLE:
Lateral torsional buckling resistance (Example 6.8 page 74).

SPECIFICATION:
Simply supported primary beam supports two secondary beams, represented with the concentrated load as shown below. The secondary beams create full lateral restraint of the primary beam web at these points. Section UB 762x267x173 is checked in grade S275 steel. The loads given are at the ultimate limit state.

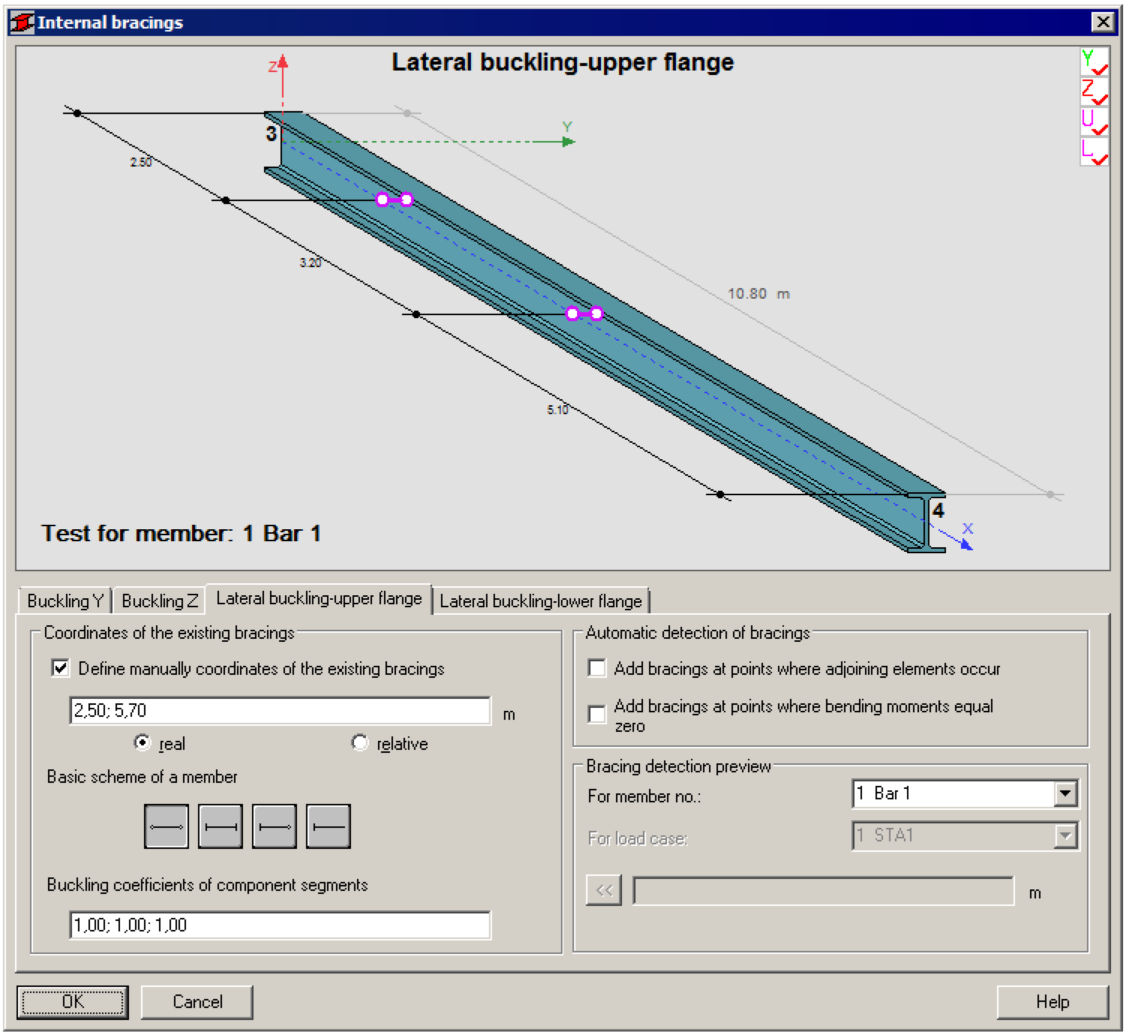


SOLUTION:
Define a new type of member. For analyzed member pre-defined type of member BEAM may be initially opened. It can be set in *Member type* combo-box. Press the *Parameters* button in DEFINITION-MEMBERS tab, which opens MEMBER DEFINITION – PARAMETERS dialog. Type a new name **Beam 1** in the *Member Type* editable field.
Select the radio button *General method (6.3.2.2.)* in the *Lateral buckling parameters.*

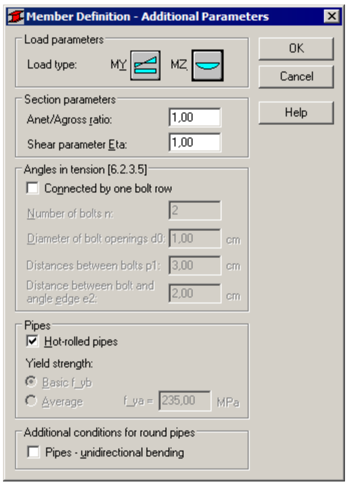


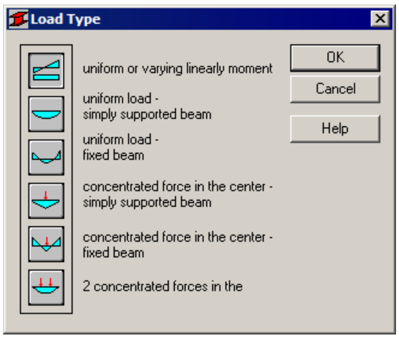
Then, press *Lateral buckling coefficient – Upper flange* icon and select the last icon (*Intermediate bracing*) that opens *Internal bracing* dialog. Define the coordinates of the existing bracing, change to *real* length radio button, type in: *2.50 5,70* (m) in the *Coordinate of the existing bracing* edit box. Close dialog by pressing OK. Do not change lateral buckling length for the lower flange.

AUTODESK



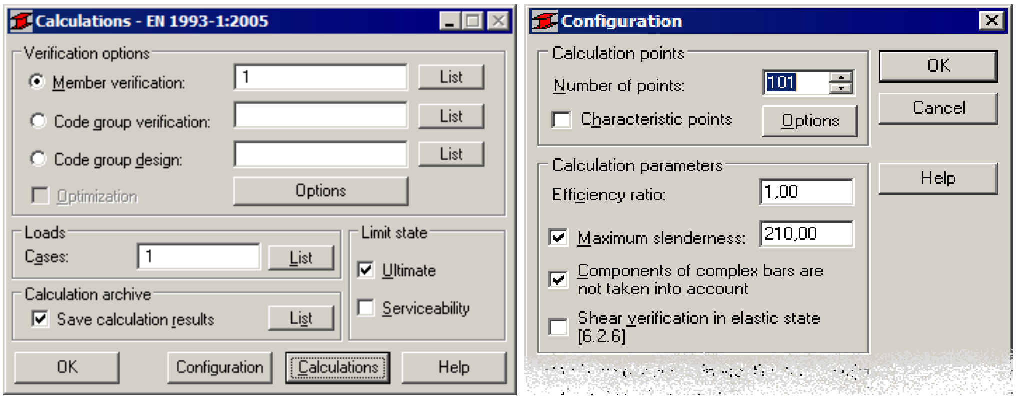
For defining appropriate load type diagram, press *More* button. Choose the icon for Load type Y and double-click the first icon (*Uniform moment and varying linearly*) in *Load Type* dialog.





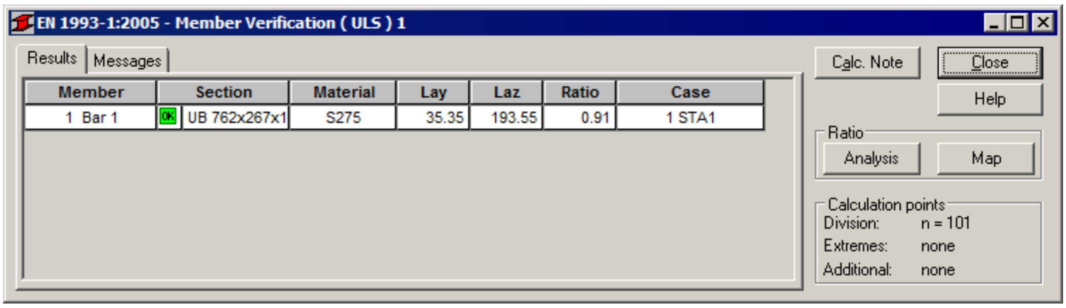
Save the newly-created type of member.
In the CALCULATIONS dialog set *Member Verification* option for member 1 and switch off *Limit State – Serviceability* (only Ultimate Limit state will be analyzed). Call configuration dialog and set number of calculation points to 101.

AUTODESK

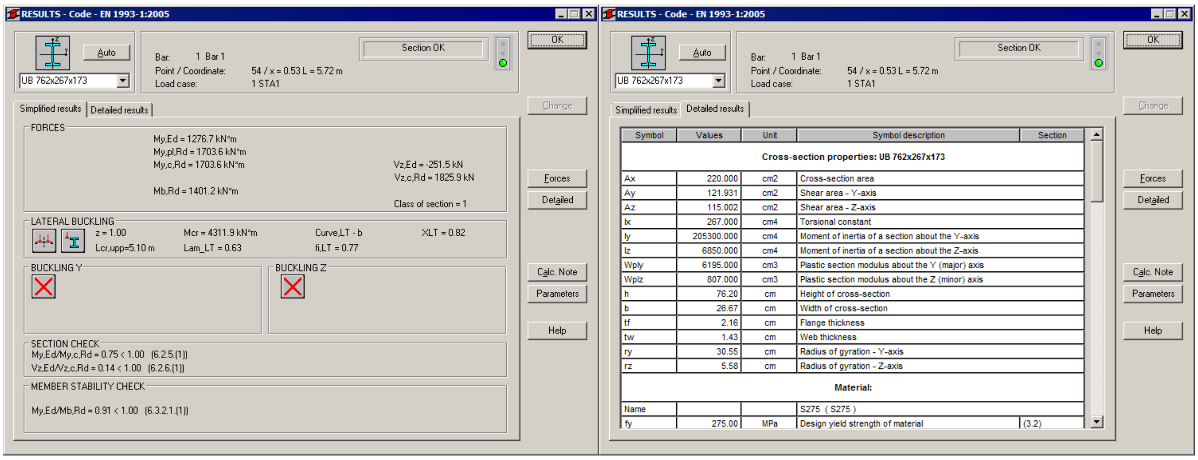


Now, start the calculations by pressing *Calculations* button.

Member Verification dialog with most significant results data will appear on screen. Pressing the line with results for member 1 opens the RESULTS dialog with detailed results for the analyzed member.



The view of the RESULTS dialog is presented below. Moreover, the printout note containing the same results data as in Simplified results tab of the RESULTS dialog is added.



STEEL DESIGN

CODE: *EN 1993-1:2005, Eurocode 3: Design of steel structures.*
ANALYSIS TYPE: Member Verification

CODE GROUP:
MEMBER: 1 Bar 1 **POINT:** 54 **COORDINATE:** x = 0.53 L = 5.72 m

---

LOADS:
Governing Load Case: 1 STA1

MATERIAL:
S275 (S275) fy = 275.00 MPa

SECTION PARAMETERS: UB 762x267x173

h=76.20 cm	gM0=1.00	gM1=1.00	
b=26.67 cm	Ay=121.931 cm2	Az=115.002 cm2	Ax=220.000 cm2
tw=1.43 cm	Iy=205300.000 cm4	Iz=6850.000 cm4	Ix=267.000 cm4
tf=2.16 cm	Wply=6195.000 cm3	Wplz=807.000 cm3	

INTERNAL FORCES AND CAPACITIES:

My,Ed = 1276.7 kN*m
My,pl,Rd = 1703.6 kN*m
My,c,Rd = 1703.6 kN*m
Vz,Ed = -251.5 kN
Vz,c,Rd = 1825.9 kN
Mb,Rd = 1401.2 kN*m
Class of section = 1

LATERAL BUCKLING PARAMETERS:

z = 1.00	Mcr = 4311.9 kN*m	Curve,LT - b	XLT = 0.82
Lcr,upp=5.10 m	Lam_LT = 0.63	fi,LT = 0.77	

BUCKLING PARAMETERS:

About Y axis: About Z axis:

VERIFICATION FORMULAS:

Section strength check:
My,Ed/My,c,Rd = 0.75 < 1.00 (6.2.5.(1))
Vz,Ed/Vz,c,Rd = 0.14 < 1.00 (6.2.6.(1))
Global stability check of member:
My,Ed/Mb,Rd = 0.91 < 1.00 (6.3.2.1.(1))

Section OK !!!

COMPARISON: Critical segment CD

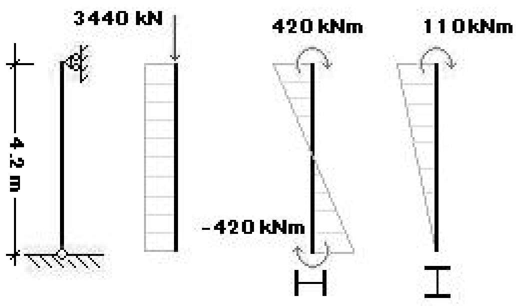
Resistance, interaction expression	Robot	Handbook
1. Critical moment for lateral-torsional buckling Mcr	4311,9	4311
2. Reduction factor for lateral-torsional buckling X_{LT}	0,82	0,82

VERIFICATION EXAMPLE 5
- Combined bi-axial bending and compression

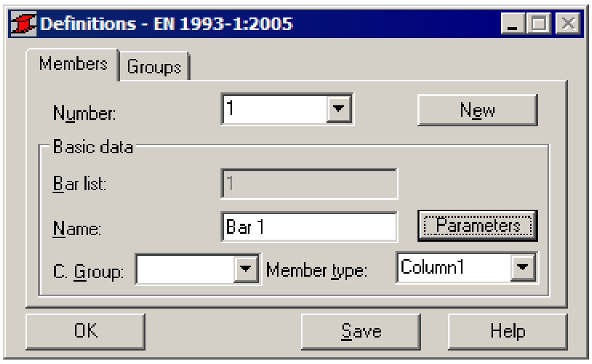
Example taken from Designer's Guide to EN 1993-1-1
L.Gardner and D.A.Nethercot, Thomas Telford Publishing, 2005

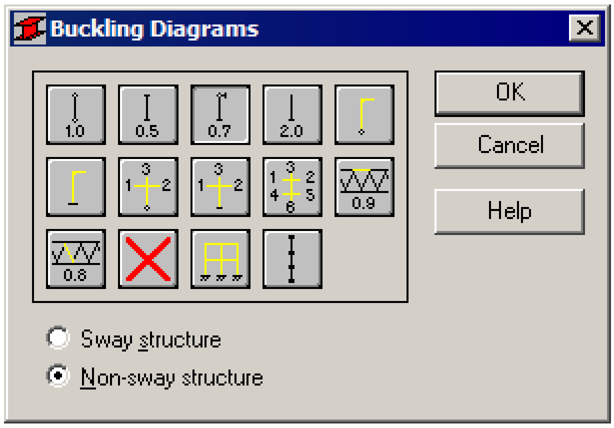
TITLE:
Combined bi-axial bending and compression (Example 6.10 page 89).

SPECIFICATION:
The model represents a column in a multistory building. The column frame is moment resisting in-plane and pinned out-of-plane, with diagonal bracing in both directions. The modeled bar shown below is pin ended about y-y and z-z axes. The bar is subjected to the compressive force and bending in major axis due to horizontal forces, in minor axis due to eccentric axial load. Section H 305x305x240 is checked in grade S275 steel. The loads are given at ultimate limit state.

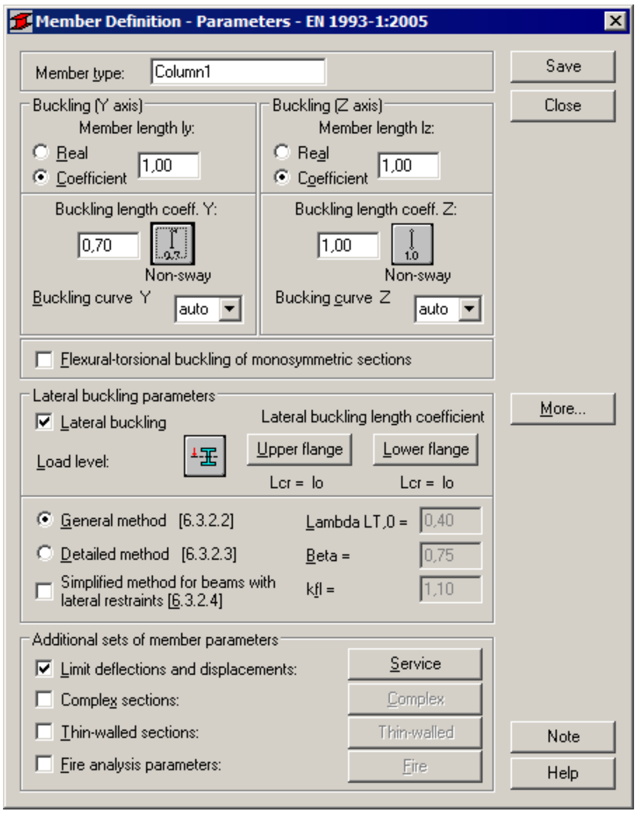


SOLUTION:
Define a new type of member. For analyzed member pre-defined type of member COLUMN may be initially opened. Press the *Parameters* button in DEFINITIONS/MEMBERS tab, which opens MEMBER DEFINITION – PARAMETERS dialog. Type a new name **Column 1** in the *Member Type* editable field. Then, press *Buckling Length coefficient Y* icon and select the third icon (*0.7*). For *Z* direction let it defined default *1.0*.



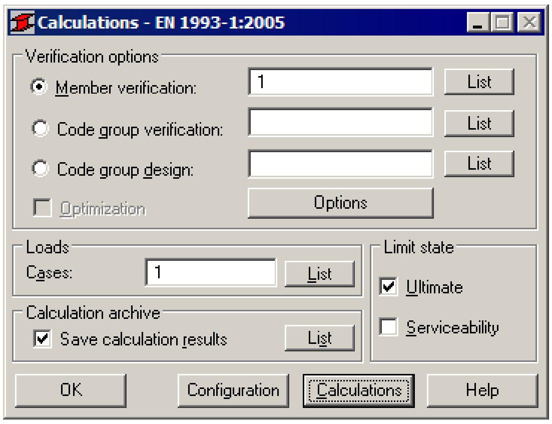


Set *Lateral buckling* checkbox.
Select the radio button *General method (6.3.2.2.)* in the *Lateral buckling parameters.*



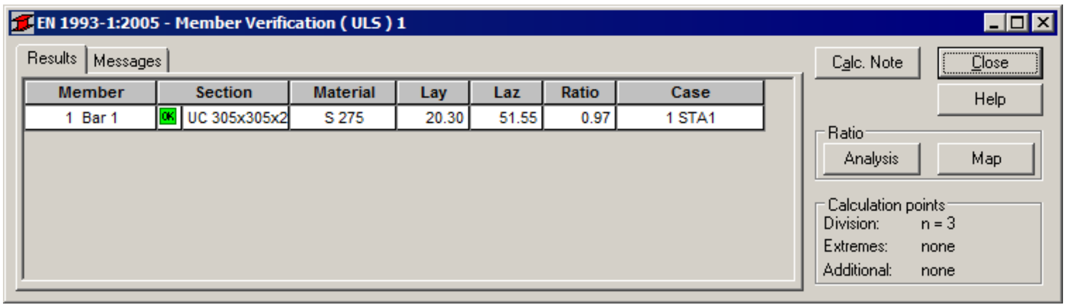
Save the newly-created type of member.

In the CALCULATIONS dialog set *Member Verification* option for member 1 and switch off *Limit State – Serviceability* (only Ultimate Limit state will be analyzed). Now, start the calculations by pressing *Calculations* button.

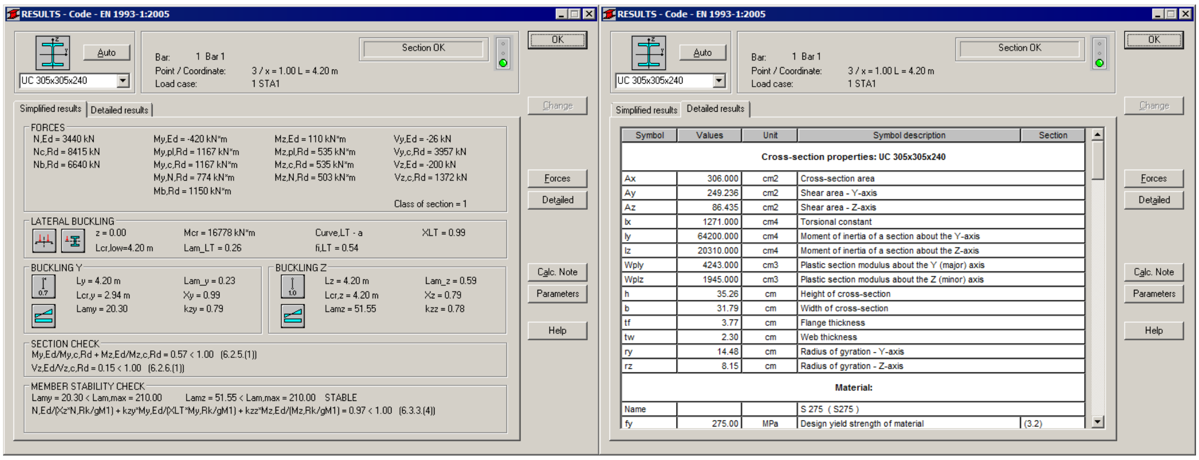


Member Verification dialog with most significant results data will appear on screen. Pressing the line with results for member 1 opens the RESULTS dialog with detailed results for the analyzed member.

AUTODESK



The view of the RESULTS dialog is presented below. Moreover, the printout note containing the same results data as in *Simplified results* tab of the RESULTS dialog is added.



STEEL DESIGN

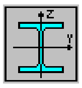
CODE: *EN 1993-1:2005, Eurocode 3: Design of steel structures.*
ANALYSIS TYPE: Member Verification

CODE GROUP:
MEMBER: 1 Bar 1 **POINT:** 3 **COORDINATE:** x = 1.00 L = 4.20 m

----

LOADS:
Governing Load Case: 1 STA1

MATERIAL:
S 275 (S275) fy = 275.00 MPa



SECTION PARAMETERS: UC 305x305x240

h=35.26 cm	gM0=1.00	gM1=1.00	
b=31.79 cm	Ay=249.236 cm2	Az=86.435 cm2	Ax=306.000 cm2
tw=2.30 cm	Iy=64200.000 cm4	Iz=20310.000 cm4	Ix=1271.000 cm4
tf=3.77 cm	Wply=4243.000 cm3	Wplz=1945.000 cm3	

INTERNAL FORCES AND CAPACITIES:

N,Ed = 3440 kN	My,Ed = -420 kN*m	Mz,Ed = 110 kN*m	Vy,Ed = -26 kN
Nc,Rd = 8415 kN	My,pl,Rd = 1167 kN*m	Mz,pl,Rd = 535 kN*m	Vy,c,Rd = 3957 kN
Nb,Rd = 6640 kN	My,c,Rd = 1167 kN*m	Mz,c,Rd = 535 kN*m	Vz,Ed = -200 kN
	My,N,Rd = 774 kN*m	Mz,N,Rd = 503 kN*m	Vz,c,Rd = 1372 kN
	Mb,Rd = 1150 kN*m		
			Class of section = 1

LATERAL BUCKLING PARAMETERS:

z = 0.00 Mcr = 16778 kN*m Curve,LT - a XLT = 0.99
Lcr,low=4.20 m Lam_LT = 0.26 fi,LT = 0.54

BUCKLING PARAMETERS:

About Y axis:		About Z axis:	
Ly = 4.20 m	Lam_y = 0.23	Lz = 4.20 m	Lam_z = 0.59
Lcr,y = 2.94 m	Xy = 0.99	Lcr,z = 4.20 m	Xz = 0.79
Lamy = 20.30	kzy = 0.79	Lamz = 51.55	kzz = 0.78

VERIFICATION FORMULAS:

Section strength check:

N,Ed/Nc,Rd = 0.41 < 1.00 (6.2.4.(1))
My,Ed/My,c,Rd + Mz,Ed/Mz,c,Rd = 0.57 < 1.00 (6.2.5.(1))
(My,Ed/My,N,Rd)^ 2.00 + (Mz,Ed/Mz,N,Rd)^2.04 = 0.34 < 1.00 (6.2.9.1.(6))
Vy,Ed/Vy,c,Rd = 0.01 < 1.00 (6.2.6.(1))
Vz,Ed/Vz,c,Rd = 0.15 < 1.00 (6.2.6.(1))

Global stability check of member:

Lambda,y = 20.30 < Lambda,max = 210.00 Lambda,z = 51.55 < Lambda,max = 210.00 STABLE
My,Ed/Mb,Rd = 0.37 < 1.00 (6.3.2.1.(1))
N,Ed/(Xy*N,Rk/gM1) + kyy*My,Ed/(XLT*My,Rk/gM1) + kyz*Mz,Ed/(Mz,Rk/gM1) = 0.66 < 1.00 (6.3.3.(4))
N,Ed/(Xz*N,Rk/gM1) + kzy*My,Ed/(XLT*My,Rk/gM1) + kzz*Mz,Ed/(Mz,Rk/gM1) = 0.97 < 1.00 (6.3.3.(4))

Section OK !!!

COMPARISON:

Resistance, interaction expression	Robot	Handbook
1. Cross section check for bi-axial bending (6.2.9.1.(6))	0,34	0,33
2. Lateral torsion buckling resistance (6.3.2.1.(1))	0,36	0,36
3. Interaction formuales (6.3.3.(4))	0,66	0,66
4. Interaction formuales (6.3.3.(4))	0,97	0,97

CONCRETE

1. Eurocode 2 EN 1992-1-1:2004 AC:2008 - RC beams

VERIFICATION EXAMPLE 1
- Dimensioning reinforcement in rectangular section at bending

Example based on:
[1] Bases of designing of reinforced and prestressed concrete structures according to Eurocode 2 (in Polish). Dolnośląskie Wydawnictwa Edukacyjne, Wroclaw 2006, Example 6.7, pp. 319 *

*** -** NOTE: the reference handbook [1] is based on the edition of Eurocode 2 EN1992-1-1:2004. The calculations in program Autodesk Robot Structural Analysis Professional are based on the last edition with the corrections EN1992-1-1:2004 AC:2008 made in 2008. However, the last correction does not introduce any changes within the range of the calculations presented in the examples.

DESCRIPTION OF THE EXAMPLE:

Calculate the reinforcement in rectangular section at simple bending at ULS. In this example, the results of the program are compared against [1]. One should note that we deal with theoretical (required) areas of reinforcement here. The real (provided) reinforcement is generated by the program in order to fulfill the theoretical reinforcement requirements and structural requirements, and is not analyzed here.

GEOMETRY:

cross section:	30x45	[cm]
cover to axis of longitudinal bars:	c = 4	[cm]

MATERIAL:

Concrete:	C25/30	
α_{cc} = 0.85		
Steel:	fyk=355	[MPa]

LOADS:

Bending moment M = 100kNm [cm^2]

IMPORTANT STEPS:

Define the geometry of the beam (*Fig.1.1*). The span geometry and the loads should be defined in order to obtain bending moment in the mid-span equal to 100 kNm *(Fig.1.2).* Set proper concrete (C25/30 with parabolic-rectangular model) and steel with fyk=355MPa (18G2) in *Calculation Options*. In order to select steel different than available by default for EN1992-1-1 code (i.e. with fyk=355MPa) which is used in [1], select PN_2002# database in *Job Preferences/Databases/Reinforcing bars (Fig.1.3)*. The authors of [1] use the partial factor α_{cc} = 0.85. The default value for the general edition of the code is α_{cc} = 1.0. In order to enable the comparison, change the factor to 0.85 in *Job Preferences/Design Codes/Partial factors for a Code EN 1992-1-1:2004 AC:2008/User defined (Fig.1.4).*

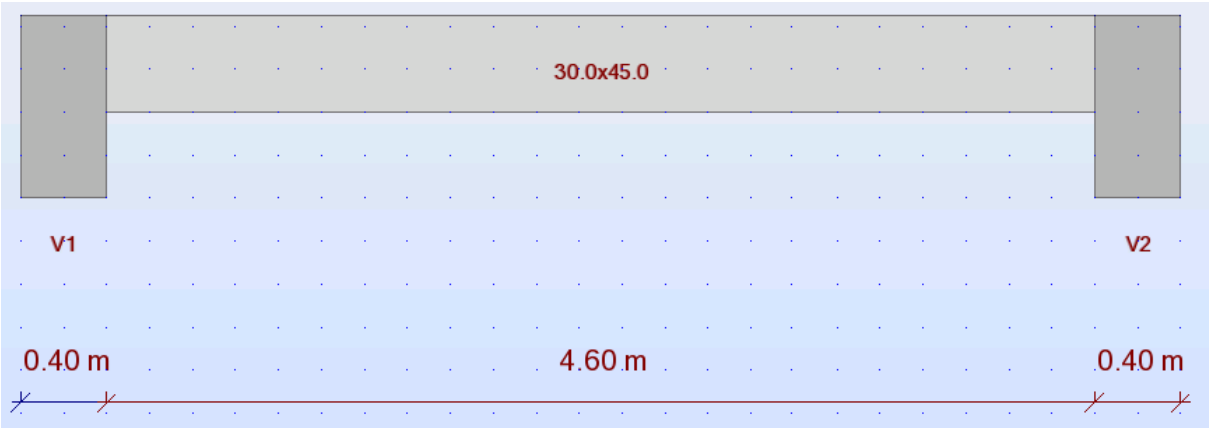
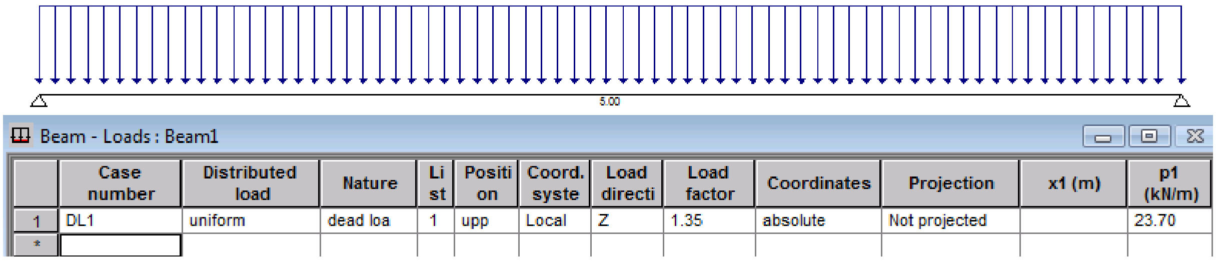


Fig. 1.1 Beam geometry



	Case number	Distributed load	Nature	List	Position	Coord. syste	Load directi	Load factor	Coordinates	Projection	x1 (m)	p1 (kN/m)
1	DL1	uniform	dead loa	1	upp	Local	Z	1.35	absolute	Not projected		23.70
*												

Fig. 1.2 Loads and the calculation model

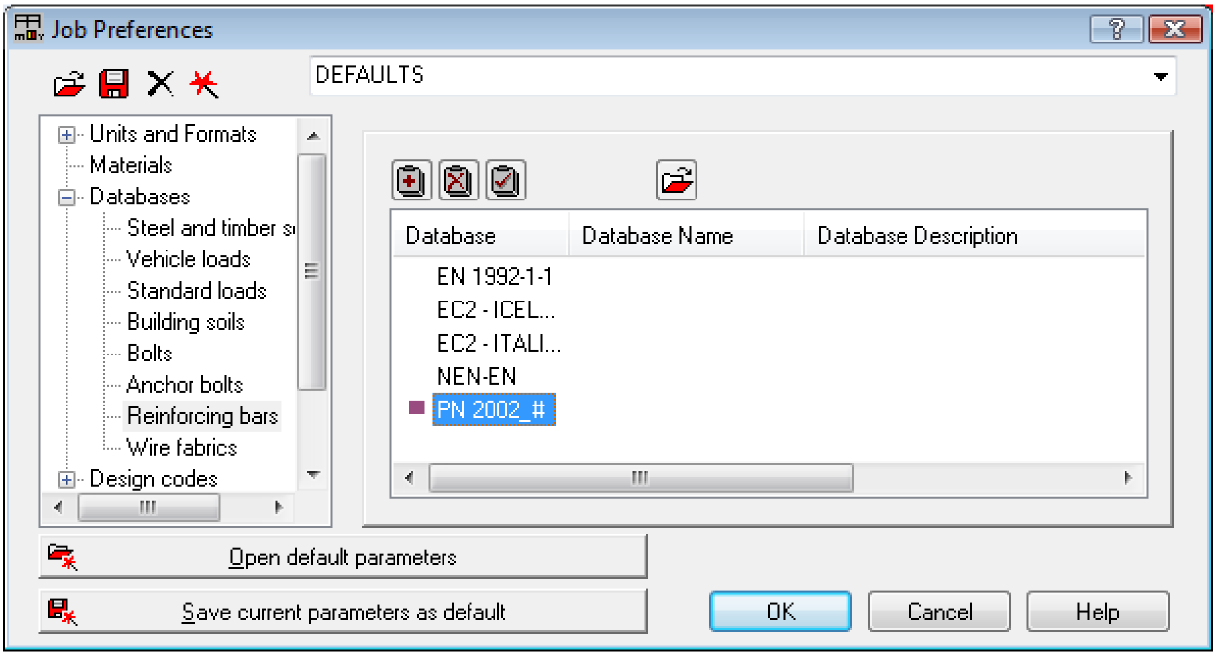


Fig. 1.3 Selection of steel database corresponding to [1]

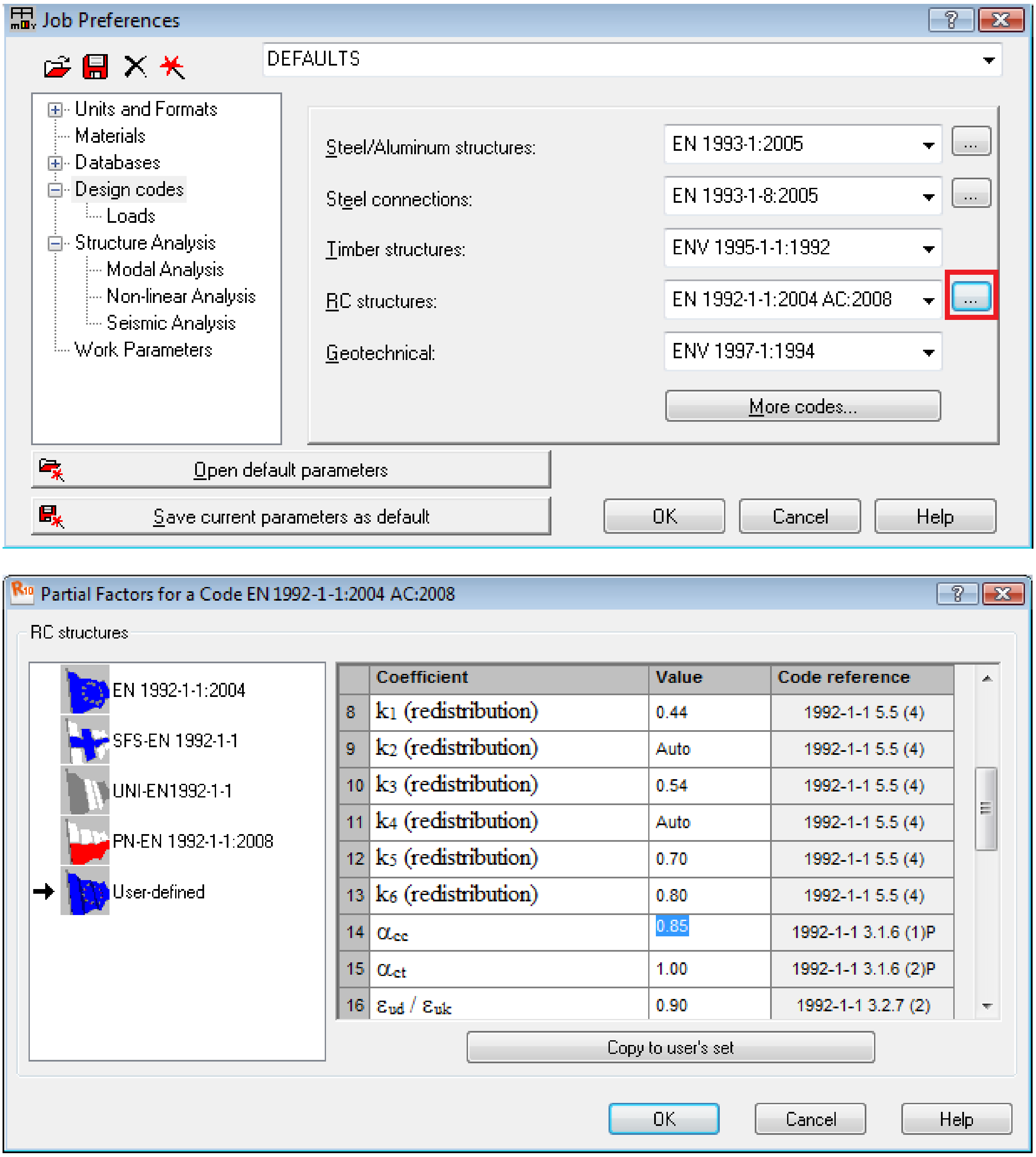


Fig. 1.4 Definition of partial factors

RESULTS OF LONGITUDINAL REINFORCEMENT (REINFORCEMENT FOR BENDING) CALCULATION:

The theoretical areas of reinforcement determined by the program are presented on the graph in *Fig.1.5.* The value in the midspan, compared with [1], is presented in the table below.

Theoretical areas	[1]	Robot
bottom reinforcement A_{s1}	8.53 cm^2	8.57 cm^2

As can be seen, very good agreement of the results is obtained.

AUTODESK

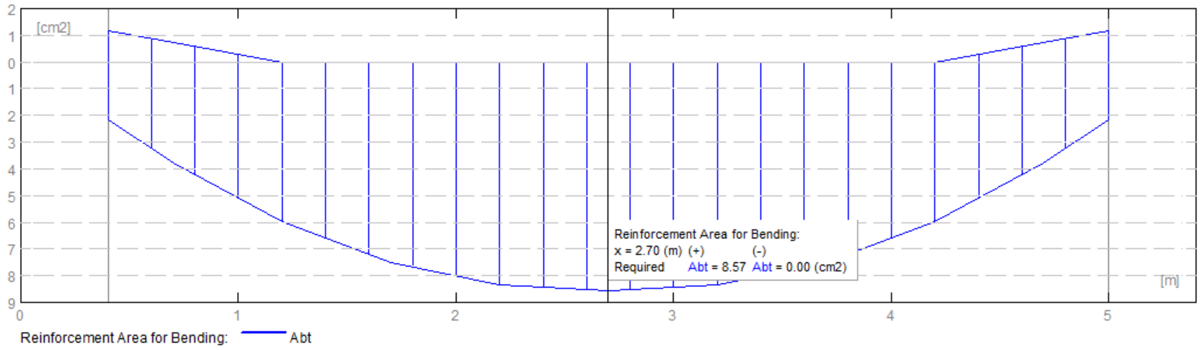


Fig. 1.5. Theoretical (required) areas of reinforcement in beam.

ANALYSIS OF RESULTS FOR NADs:

The example presented here has been calculated for the parameters assumed in [1]. As discussed above, although the example is calculated for general edition of the code [2], the authors of [1] use the partial factor α_{cc} = 0.85. The default value for the general edition of the code is α_{cc} = 1.0.
In this section, the same example is calculated for different national editions of Eurocode 2. The results of calculations are compared in the table below, along with the values of coefficients, which allow you to understand the possible differences of reinforcement area for different NADs.

Code	γ_c	γ_s	α_{cc}	bottom reinf. A_{s1}-Robot results
Handbook example (general Eurocode 2 edition with modified α_{cc})	1.5	1.15	0.85	8.57 cm^2
EN 1992-1-1:2004 AC:2008	1.5	1.15	1.0	8.45 cm^2
PN-EN 1992-1-1:2008	1.4	1.15	1.0	8.41 cm^2
UNI-EN 1992-1-1	1.5	1.15	0.85	8.57 cm^2
SFS-EN 1992-1-1	1.5	1.15	0.85	8.57 cm^2
EN 1992-1-1 DK NA:2007	1.45	1.2	1.0	8.80 cm^2
BS EN1992-1-1:2004 NA2005	1.5	1.15	0.85	8.57 cm^2
NS-EN 1992-1-1:2004/NA:2008	1.5	1.15	0.85	8.57 cm^2
NF EN 1992-1-1/NA:2007	1.5	1.15	1.0	8.45 cm^2

As it can be seen above, the results may slightly differ for some NADs due to different material coefficients. However, the manual calculations carried out show that the results are correct for all cases.

VERIFICATION EXAMPLE 2
- Dimensioning reinforcement in rectangular section at bending

Example based on:
[1] Bases of designing of reinforced and prestressed concrete structures according to Eurocode 2 (in Polish). Dolnośląskie Wydawnictwa Edukacyjne, Wroclaw 2006, Example 6.8, pp. 330*

*** -** NOTE: the reference handbook [1] is based on the edition of Eurocode 2 EN1992-1-1:2004. The calculations in program Autodesk Robot Structural Analysis Professional are based on the last edition with corrections EN1992-1-1:2004 AC:2008. However, the last correction does not introduce any changes within the range of the calculations presented in the examples.

DESCRIPTION OF THE EXAMPLE:

Calculate the reinforcement in rectangular section at simple bending at ULS. In this example, the results of the program are compared against [1]. The data is the same as in Verification problem 1, except for the bending moment which is equal to M=320 kNm.

RESULTS OF LONGITUDINAL REINFORCEMENT (REINFORCEMENT FOR BENDING) CALCULATION:

The theoretical areas of reinforcement determined by the program are presented on the graph in *Fig.2.1.* The values in the midspan, compared with [1], are presented in the table below.

Theoretical areas	[1]	Robot
bottom reinf. A_{s1}	34.59 cm^2	34.62 cm^2
top reinf. A_{s2}	2.98 cm^2	2.91 cm^2

As can be seen, very good agreement of the results is obtained.

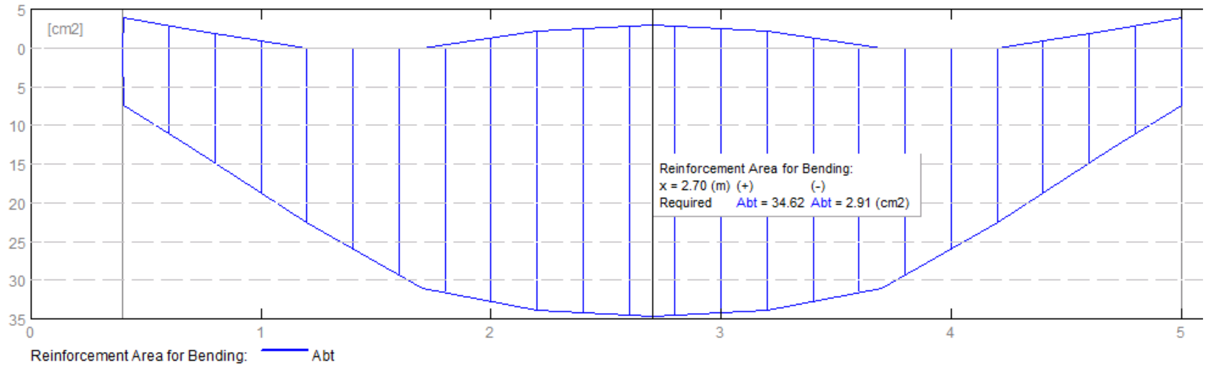


Fig. 2.1. Theoretical (required) areas of reinforcement in beam.

ANALYSIS OF RESULTS FOR NADs:

The presented example has been calculated for the parameters assumed in [1]. As discussed above, although the example is calculated for general edition of the code [2], the authors of [1] use the partial factor α_{cc} = 0.85. The default value for the general edition of the code is α_{cc} = 1.0.
In this section, the same example is calculated for different national editions of Eurocode 2. The results of calculations are compared in the table below, along with the values of coefficients which allow you to understand the possible differences for different NADs.

AUTODESK

Code	γ_c	γ_s	α_{cc}	bottom reinf. A_{s1}- Robot results	top reinf. A_{s2}- Robot results
Handbook example (general Eurocode 2 edition with modified α_{cc})	1.5	1.15	0.85	34.73 cm^2	2.92 cm^2
EN 1992-1-1:2004 AC:2008	1.5	1.15	1.0	34.27 cm^2	0.0 cm^2
PN-EN 1992-1-1:2008	1.4	1.15	1.0	33.09 cm^2	0.0 cm^2
UNI-EN 1992-1-1	1.5	1.15	0.85	34.73 cm^2	2.92 cm^2
SFS-EN 1992-1-1	1.5	1.15	0.85	34.73 cm^2	2.92 cm^2
EN 1992-1-1 DK NA:2007	1.45	1.2	1.0	35.12 cm^2	0.0 cm^2
BS EN1992-1-1:2004 NA2005	1.5	1.15	0.85	34.73 cm^2	2.92 cm^2
NS-EN 1992-1-1:2004/NA:2008	1.5	1.15	0.85	34.73 cm^2	2.92 cm^2
NF EN 1992-1-1/NA:2007	1.5	1.15	1.0	34.27 cm^2	0.0 cm^2

As it can be seen above, the results may slightly differ for some NADs due to the different material coefficients. However, the manual calculations carried out show that the results are correct for all cases.

VERIFICATION EXAMPLE 3
- Dimensioning reinforcement in rectangular section at bending with compression

Example based on:
[1] Bases of designing of reinforced and prestressed concrete structures according to Eurocode 2 (in Polish). Dolnośląskie Wydawnictwa Edukacyjne, Wroclaw 2006, Example 6.9, pp. 333*

*** -** NOTE: the reference handbook [1] is based on the edition of Eurocode 2 EN1992-1-1:2004. The calculations in program Autodesk Robot Structural Analysis Professional are based on the last edition with the corrections EN1992-1-1:2004 AC:2008. However, the last correction does not introduce any changes within the range of the calculations presented in the examples.

DESCRIPTION OF THE EXAMPLE:

Calculate the reinforcement in rectangular section at bending with compression at ULS. In this example, the results of the program are compared against [1]. The data is the same as in Verification problem 1, except of the forces which are: bending moment M=150 kNm, and compressive force N=150 kNm.

RESULTS OF LONGITUDINAL REINFORCEMENT (REINFORCEMENT FOR BENDING) CALCULATION:

The theoretical areas of reinforcement determined by the program are presented on the graph in *Fig.3.1.* The values in the midspan, compared with [1], are presented in the table below.

Theoretical areas	[1]	Robot
bottom reinf. A_{s1}	11.62 cm^2	11.67 cm^2

As it can be seen above, very good agreement of the results is obtained.

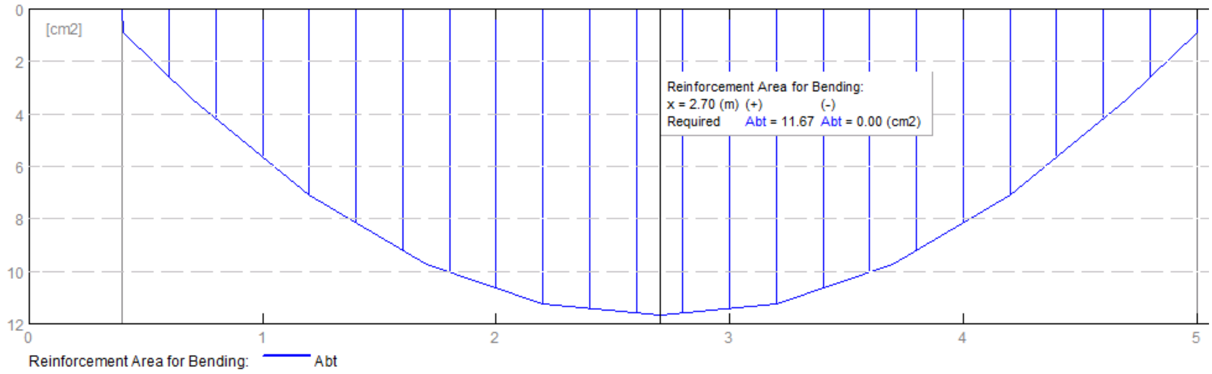


Fig. 3.1. Theoretical (required) areas of reinforcement in beam.

ANALYSIS OF RESULTS FOR NAD's:

The presented example has been calculated for the parameters assumed in [1]. As discussed above, although the example is calculated for general edition of the code [2], the authors of [1] use the partial factor α_{cc} = 0.85. The default value for the general edition of the code is α_{cc} = 1.0. In this section, the same example is calculated for different national editions of Eurocode 2. The results of calculation are compared in the table below, along with the values of coefficients, which allows you to understand the possible differences for different NADs.

Code	γ_c	γ_s	α_{cc}	bottom reinf. A_{s1}-Robot results
Handbook example (general Eurocode 2 edition with modified α_{cc})	1.5	1.15	0.85	11.67 cm^2
EN 1992-1-1:2004 AC:2008	1.5	1.15	1.0	11.17 cm^2
PN-EN 1992-1-1:2008	1.4	1.15	1.0	11.00 cm^2
UNI-EN 1992-1-1	1.5	1.15	0.85	11.67 cm^2
SFS-EN 1992-1-1	1.5	1.15	0.85	11.67 cm^2
EN 1992-1-1 DK NA:2007	1.45	1.2	1.0	11.57 cm^2
BS EN1992-1-1:2004 NA2005	1.5	1.15	0.85	11.67 cm^2
NS-EN 1992-1-1:2004/NA:2008	1.5	1.15	0.85	11.67 cm^2
NF EN 1992-1-1/NA:2007	1.5	1.15	1.0	11.17 cm^2

As it can be seen above, the results may slightly differ for some NADs due to the different material coefficients. However, the manual calculations carried out show that the results are correct for all cases.

VERIFICATION EXAMPLE 4
- Dimensioning reinforcement in rectangular section at bending with compression

Example based on:
[1] Bases of designing of reinforced and prestressed concrete structures according to Eurocode 2 (in Polish). Dolnośląskie Wydawnictwa Edukacyjne, Wroclaw 2006, Example 6.10, pp. 334 *

*** -** NOTE: the reference handbook [1] is based on the edition of Eurocode 2 EN1992-1-1:2004 from year 2004. The calculations in program Autodesk Robot Structural Analysis Professional are based on the last edition with the corrections EN1992-1-1:2004 AC:2008 from year 2008. However, the last correction does not introduce any changes within the range of the calculations presented in the examples.

DESCRIPTION OF THE EXAMPLE:

Calculate the reinforcement in rectangular section at bending with compression at ULS. In this example, the results of the program are compared against [1]. The data is the same as in Verification problem 1, except of the forces which are: bending moment M=150 kNm, and compressive force N=1000 kNm.

RESULTS OF LONGITUDINAL REINFORCEMENT (REINFORCEMENT FOR BENDING) CALCULATION:

The theoretical areas of reinforcement determined by the program are presented on the graph in *Fig.4.1.* The values in the midspan, compared with [1], are presented in the table below.

Theoretical areas	[1]	Robot
bottom reinf. A_{s1}	3.64 cm^2	3.64 cm^2
top reinf. A_{s2}	4.30 cm^2	4.34 cm^2

As can be seen, very good agreement of the results is obtained.

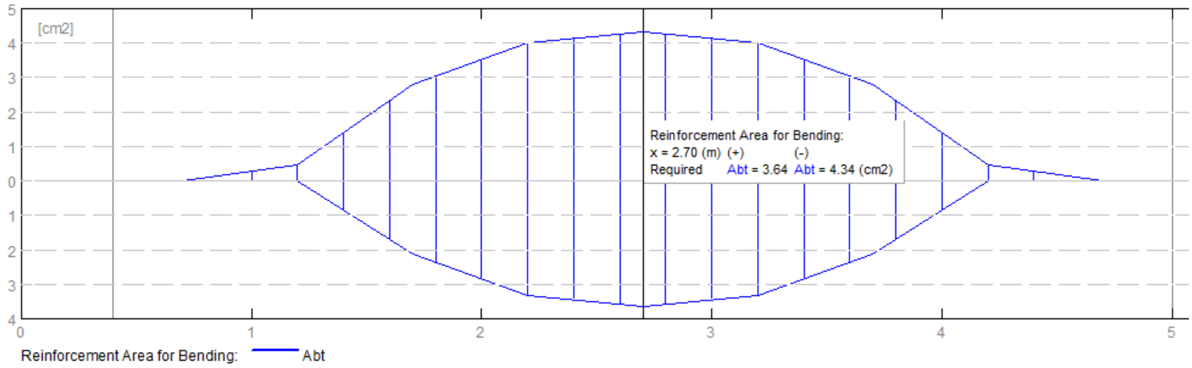


Fig. 4.1. Theoretical (required) areas of reinforcement in beam.

ANALYSIS OF RESULTS FOR NADs:

The presented example has been calculated for the parameters assumed in [1]. As discussed above, although the example is calculated for general edition of the code [2], the authors of [1] use the partial factor α_{cc} = 0.85. The default value for the general edition of the code is α_{cc} = 1.0. In this section, the same example is calculated for different national editions of the Eurocode 2. The results of calculation are compared in the table below, along with the values of coefficients which allows you to understand the possible differences for different NADs.

Code	γ_c	γ_s	α_{cc}	bottom reinf. A_{s1}- Robot results	top reinf. A_{s2}- Robot results
Handbook example (general Eurocode 2 edition with modified α_{cc})	1.5	1.15	0.85	3.64 cm^2	4.34 cm^2
EN 1992-1-1:2004 AC:2008	1.5	1.15	1.0	4.75 cm^2	0.0 cm^2
PN-EN 1992-1-1:2008	1.4	1.15	1.0	3.24 cm^2	0.0 cm^2
UNI-EN 1992-1-1	1.5	1.15	0.85	3.64 cm^2	4.34 cm^2
SFS-EN 1992-1-1	1.5	1.15	0.85	3.64 cm^2	4.34 cm^2
EN 1992-1-1 DK NA:2007	1.45	1.2	1.0	4.13 cm^2	0.0 cm^2
BS EN1992-1-1:2004 NA2005	1.5	1.15	0.85	3.64 cm^2	4.34 cm^2
NS-EN 1992-1-1:2004/NA:2008	1.5	1.15	0.85	3.64 cm^2	4.34 cm^2
NF EN 1992-1-1/NA:2007	1.5	1.15	1.0	4.75 cm^2	0.0 cm^2

As it can be seen above, the results may slightly differ for some NADs due to the different material coefficients. However, the manual calculations carried out show the results are correct for all cases.

VERIFICATION EXAMPLE 5
- Dimensioning of shear reinforcement in beam with rectangular section

Example based on:
Manual calculations according to:
[2] Eurocode 2 EN 1992-1-1:2004 AC:2008, point 6.2

DESCRIPTION OF THE EXAMPLE:

Calculate the shear reinforcement in simply supported beam with rectangular section. In this example, the results of the program are compared against the manual calculations presented.

GEOMETRY:

cross section:	30x45	[cm]
cover to axis of longitudinal bars:	c = 4	[cm]

MATERIAL:

Concrete:	C20/255
Steel:	B500C (f_{yk} = 500 [MPa])

LOADS:

Uniformly distributed:

Dead load:	q_D = 30 [kN/m]
Live load:	q_L = 20 [kN/m]

IMPORTANT STEPS:

Define the geometry of the beam (*Fig.5.1*) and loads (*Fig.5.2*). Set proper concrete and steel in *Calculation Options*. Set allowable stirrups spacings to: 0.05; 0.07; 0.10; 0.20; 0.25; 0.30; 0.35; 0.40; 0.50.

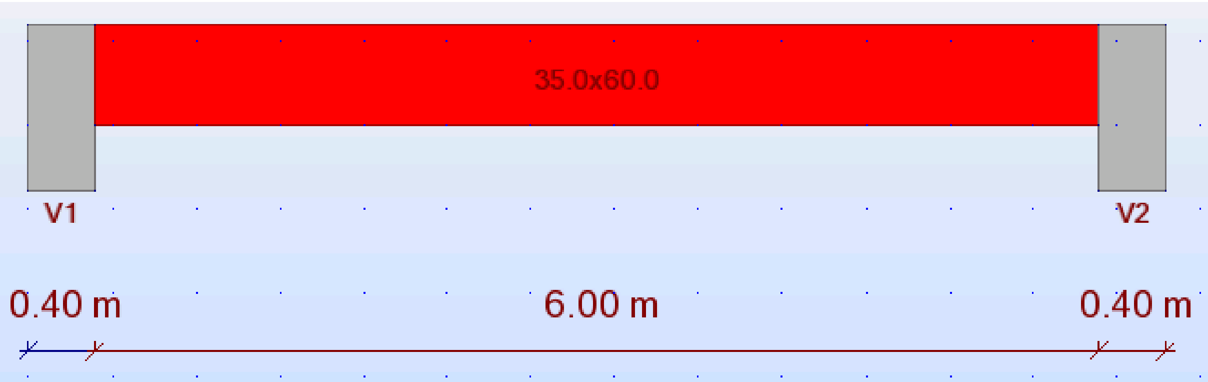
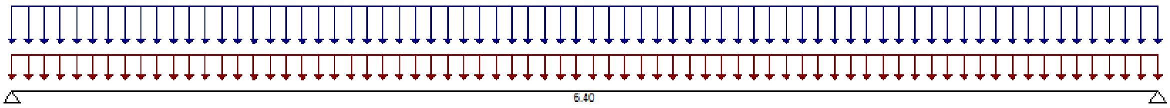


Fig. 5.1 Beam geometry



Beam - Loads : Beam1

	Case number	Distributed load	Nature	List	Position	Coord. syste	Load directi	Load factor	Coordinates	Projection	x1 (m)	p1 (kN/m)
1	DL1	uniform	dead loa	1	upp	Local	Z	1.35	absolute	Not projected		30.00
2	LL1	uniform	Live	1	upp	Local	Z	1.50	absolute	Not projected		20.00
*												

Fig. 5.2 Loads and the calculation model

RESULTS OF SHEAR REINFORCEMENT DIMENSIONING:

- **CALCULATION OF MAXIMUM SHEAR FORCE:**

Load nature:	Characteristic load [kN/m]	Load factor	Design load [kN/m]
Dead load	30	1.35	40.5
Live load	20	1.5	30
		$q_{tot} =$	70.5

The shear force at the end of the beam is equal to:

$$V_{x=0} = q_{tot} \cdot \frac{l}{2} = 239.7kN$$

$$l = 6.8m$$

The shear force at the edge of the support is equal to:

$$V_{x=0.4} = V_{x=0} - q_{tot} \cdot 0.4 = 211.5kN$$

The value of shear force calculated above is in agreement with the value calculated in Robot (see *Fig. 5.3*).

- **CALCULATION OF SHEAR CAPACITY OF A BEAM WITHOUT SHEAR REINFORCEMENT:**

The shear capacity of element without shear reinforcement is calculated based on eq. (6.2.a) [2]. The shear capacity in the mid-span is:

$$V_{Rd,c} = \left[C_{Rd,c} k (100 \rho_l f_{ck})^{1/3} + k_1 \sigma_{cp}\right] b_w d = 103.69kN$$

$$C_{Rd,c} = 0.18 / \gamma_c = 0.12$$

$$k = 1 + \sqrt{200/d} = 1.61 \le 2.0$$

$$d = 600 - 65 = 535mm$$ (position of bottom bars is averaged for two layers)

$$\rho_l = \frac{A_{sl}}{b_w d} = 0.0117$$

$$A_{sl} = 2199mm^2$$

$$b_w = 350mm$$

$$f_{ck} = 20MPa$$

$$\sigma_{cp} = 0MPa$$

But should not be smaller than:

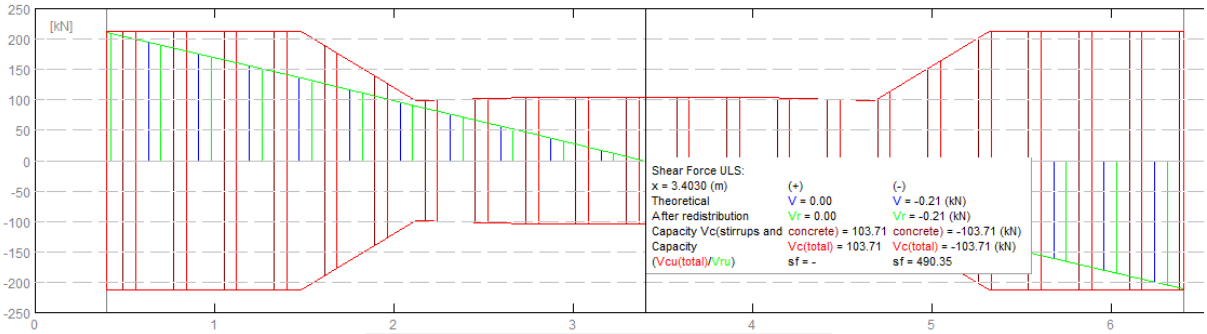
$$V_{Rd,c} = \left[v_{min} + k_1 \sigma_{cp}\right] b_w d = 59.9kN$$

$$v_{min} = 0.035 k^{3/2} f_{ck}^{1/2} = 0.32$$

AUTODESK

The value of $V_{Rd,c}$ calculated by the program is in very good agreement with the one calculated above (see table below). The value calculated by the program may be found as the shear capacity in the point where shear reinforcement is placed in maximum allowable spacings (e.g. in the mid-span) (*Fig.5.3*).

Theoretical areas	Manual calculation	Robot
Shear capacity $V_{Rd,c}$	103.69 kN	103.71 kN



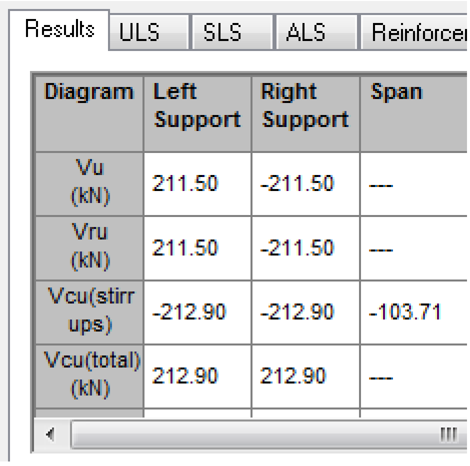


Diagram	Left Support	Right Support	Span
Vu (kN)	211.50	-211.50	---
Vru (kN)	211.50	-211.50	---
Vcu(stirrups)	-212.90	-212.90	-103.71
Vcu(total) (kN)	212.90	212.90	---

Fig. 5.3 Shear force distribution and shear capacity

- **CALCULATION OF SHEAR CAPACITY OF A BEAM WITH SHEAR REINFORCEMENT:**

Since, at the support face $V \geq V_{Rd,c}$ the shear reinforcement must be calculated. The shear reinforcement should be distributed along the length 1.4 m from the support face (see *Fig.5.3*). Using equation (6.8) [2]:

$$V_{Rd,s} = \frac{A_{sw}}{s} z f_{ywd} \cot\theta$$

And assuming $V_{Rd,s} = V_{x=0.4}$, the required spacing of stirrups near the support is:

$$s = \frac{A_{sw}}{V_{x=0.4}} z f_{ywd} \cot\theta = 0.101m$$

$$A_{sw} = 0.000101m^2$$ (2 bars ϕ8)

$$V_{x=0.4} = 211.5kN$$

$$z = 0.9d = 0.49m$$

$$d = 0.6 - 0.059 = 0.541m$$ (for bottom bars at the support)

AUTODESK

$$f_{ywd} = f_{ywk} / \gamma_s = 434.8MPa$$
$$f_{ywk} = 500MPa$$
$$\gamma_s = 1.15$$
$$\cot\theta = 1.0$$
(set in Calculation options/General)

The assumed spacing near the support is equal to 0.1 m (see *Fig.5.4*). Thus, the shear capacity is equal to:

$$V_{Rd,s} = \frac{A_{sw}}{s} z f_{ywd} \cot\theta = 212.9kN$$

And should not be greater than: $VRd,\max = \dfrac{\alpha_{cw} b_w z \nu_1 f_{cd}}{\cot\theta + \tan\theta} = 627.4kN$

$$\alpha_{cw} = 1.0$$
$$\nu_1 = 0.552$$
$$f_{cd} = f_{ck} / \gamma_c = 13.33MPa$$

The value of $V_{Rd,s}$ at the support face calculated by the program (*Fig.5.3*) is in agreement with the one calculated above (see table below).

	Manual calculation	Robot
Shear capacity $V_{Rd,s}$	212.9 kN	212.9 kN

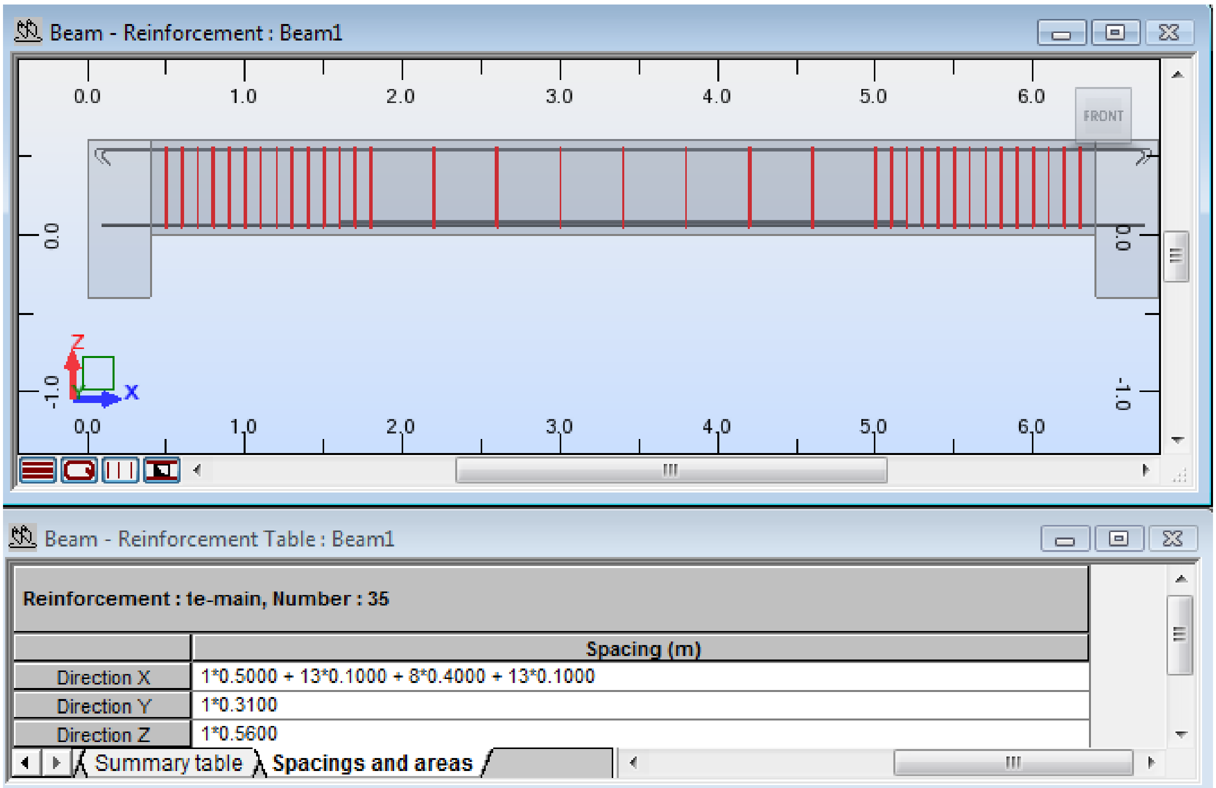


Fig. 5.4 Shear reinforcement distribution (see Direction X in the Reinforcement table)

ANALYSIS OF RESULTS FOR NADs:

The presented example has been calculated for the general edition of Eurocode 2 [2]. In this section, the same example is calculated for different national editions of the Eurocode 2. The results of calculation are compared in the table below, along with the values of coefficients which allows you to understand the possible differences for different NADs.

Code	γ_c	γ_s	α_{cc}	Shear capacity $V_{Rd,c}$	Shear capacity $V_{Rd,s}$
EN 1992-1-1:2004 AC:2008 (manual calculation)	1.5	1.15	1.0	103.71 kN	212.9 kN
PN-EN 1992-1-1:2008	1.4	1.15	1.0	111.12 kN	212.9 kN
UNI-EN 1992-1-1	1.5	1.15	0.85	103.71 kN	212.9 kN
SFS-EN 1992-1-1	1.5	1.15	0.85	103.71 kN	212.9 kN
EN 1992-1-1 DK NA:2007	1.45	1.2	1.0	107.71 kN	203.29 kN
BS EN1992-1-1:2004 NA2005	1.5	1.15	0.85	103.71 kN	236.03 kN
NS-EN 1992-1-1:2004/NA:2008	1.5	1.15	0.85	103.71*	236.03 kN
NF EN 1992-1-1/NA:2007	1.5	1.15	1.0	103.71 kN	212.13 kN

As it can be seen, the value of shear capacity $V_{Rd,s}$ is dependent upon the varying γ_c coefficient for different national editions of the code. The difference concerning the value of $V_{Rd,c}$ is due to the $C_{Rd,c}$ coefficient dependent upon γ_c.

* NOTE: The spacing of stirrups of 40cm used in other editions of the code is greater than the maximum allowable spacing according to NS-EN 1992-1-1:2004/NA:2008, thus the spacing of stirrups in the mid-span should be decreased down to 25cm.

VERIFICATION EXAMPLE 6
- Deflection of simply supported beam with rectangular section

Example based on:
[1] Bases of designing of reinforced and prestressed concrete structures according to Eurocode 2 (in Polish). Dolnośląskie Wydawnictwa Edukacyjne, Wroclaw 2006, Example 11.9.5, pp. 642 *

***** - NOTE: the reference handbook [1] is based on the edition of Eurocode 2 EN1992-1-1:2004. The calculations in program Autodesk Robot Structural Analysis Professional are based on the last edition with the corrections EN1992-1-1:2004 AC:2008. However, the last correction does not introduce any changes within the range of the calculations presented in the examples.

DESCRIPTION OF THE EXAMPLE:

Calculate the deflection of simply supported beam with rectangular section after cracking. In this example, the results of the program are compared against the results presented in [1]. However, slight modification of the example published in [1] is done for the sake of this verification. The authors of [1] calculate the deflection taking into account the influence of shrinkage. This is not the case in Robot program. In order to enable the comparison of the results, the reference value of final deflection is obtained by means of recalculation of deflection, neglecting the shrinkage effects (but using other partial results presented in [1]).

GEOMETRY:

cross section:	30x50	[cm]
cover to axis of longitudinal bars:	c = 5	[cm]
span length:	l=7.5	[m]

MATERIAL:

Concrete: C16/20

REINFORCEMENT:

Bottom bars: 5ϕ20

LOADS:

Quasi-permanent bending moment M: = 160 [kNm]

IMPORTANT STEPS:

Define the geometry of the beam (*Fig.6.1*) and loads, which lead to the bending moment at SLS equal to 160kNm in the mid-span (*Fig.6.2*). Set proper concrete in *Calculation Options*.

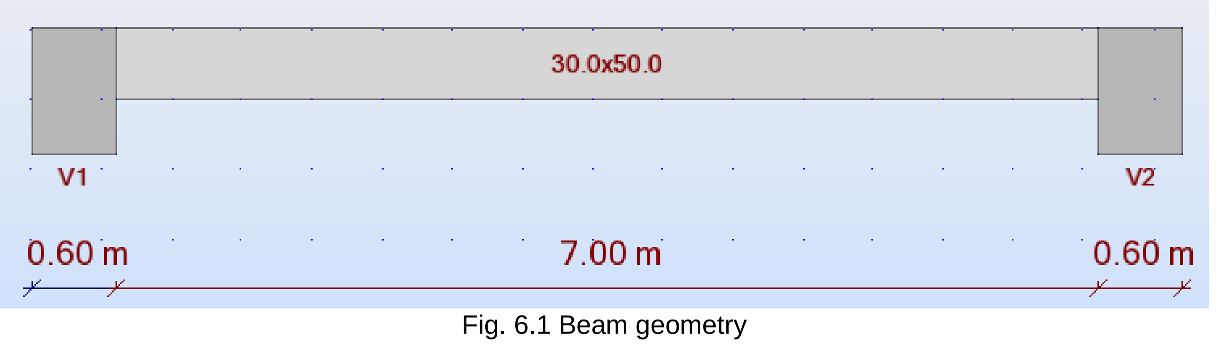
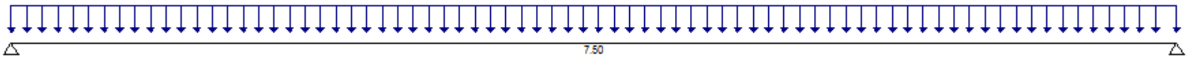


Fig. 6.1 Beam geometry



Case number	Distributed load	Nature	List	Position	Coord. syste	Load directi	Load factor	Coordinates	Projection	x1 (m)	p1 (kN/m)
DL1	uniform	dead loa	1	upp	Local	Z	1.35	absolute	Not projected		22.76

Fig. 5.2 Loads and the calculation model

NOTE: the program automatically generates reinforcement different than assumed in [1]. This is because the example in [1] concerns the SLS effects only, while Robot calculates the reinforcement for ULS and SLS (in this case, the deflection is additionally limited by the program). For the sake of only-deflection analysis, the reinforcement should be modified manually to the form as assumed in [1]. Since we analyze only deflection here, the transversal reinforcement may be deleted (*Fig.5.3*).

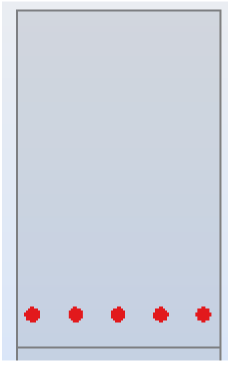


Fig. 5.2 Reinforcement (5ϕ20) assumed in [1]

RESULTS OF DEFLECTION CALCULATION:

The reference value of deflection, based on [1] after omitting shrinkage effects is:

$$f = (1-\xi) f_I + \xi f_{II} = 3.757cm$$

$$\xi = 0.9686$$

$$f_I = 2.720cm$$

$$f_{II} = 3.791cm$$

	Reference value based on [1]	Robot
Deflection f	3.757cm	3.700cm

As can be seen in the table, the results are in agreement. Slight discrepancy is a result of small difference in elastic modulus of concrete. The authors of [1] use $E_{cm} = 27500MPa$ while Robot uses the code value for C16/20 concrete, $E_{cm} = 29000MPa$.

ANALYSIS OF RESULTS FOR NADs:

The result of deflection has also been checked for national editions of Eurocode 2:

PN-EN 1992-1-1:2008
UNI-EN 1992-1-1
SFS-EN 1992-1-1
EN 1992-1-1 DK NA:2007
BS EN1992-1-1:2004 NA2005
NS-EN 1992-1-1:2004/NA:2008
NF EN 1992-1-1/NA:2007

It has been found that the results are equal for national editions and general edition [2].

LITERATURE

[1] Bases of designing of reinforced and prestressed concrete structures according to Eurocode 2 (in Polish). Dolnośląskie Wydawnictwa Edukacyjne, Wroclaw 2006

[2] Eurocode 2 EN 1992-1-1:2004 AC:2008

2. Eurocode 2 EN 1992-1-1:2004 AC:2008 - RC columns

VERIFICATION EXAMPLE 1
- Column subjected to axial load and uni-axial bending

Example based on:
[1] Bases of designing of reinforced and prestressed concrete structures according to Eurocode 2 (in Polish). Dolnośląskie Wydawnictwa Edukacyjne, Wroclaw 2006, Example 10.1, pp. 565 *

***** - NOTE: the reference handbook [1] is based on the edition of Eurocode 2 EN1992-1-1:2004. The calculations in program Autodesk Robot Structural Analysis Professional are based on the last edition with the corrections EN1992-1-1:2004 AC:2008. However, the last correction does not introduce any changes within the range of the calculations presented in the examples.

DESCRIPTION OF THE EXAMPLE:

The example illustrates the influence of second order-effects on the total moment of column AB of the frame (*Fig.1.1*). In [1], the reinforcement is assumed *a priori*. We analyze the part of the example where the total moments are determined based on two methods: the nominal curvature method and the nominal stiffness method. The total moment calculated with Robot program is verified against the results in [1] and possible differences are discussed.

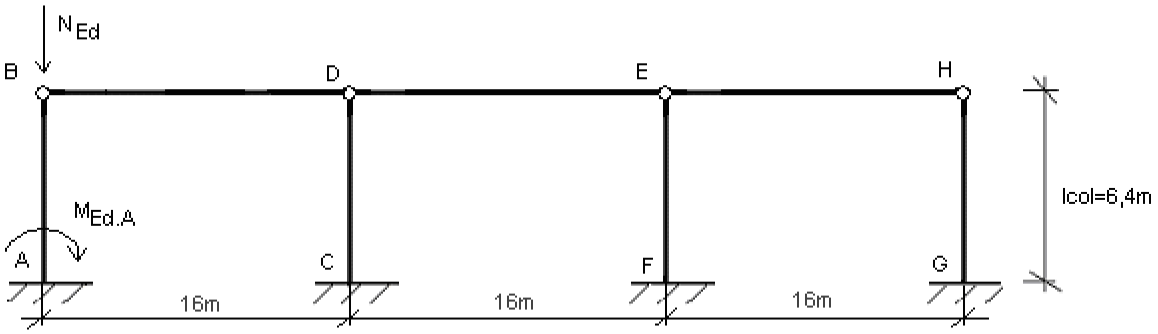


Fig. 1.1. The model of the frame with the analyzed column AB.

GEOMETRY:

cross section:	45x50 [cm]
cover to axis of longitudinal bars:	c = 3.5 [cm]
heigh of the column:	l_{col} = 6.4 [m]
number of columns in analyzed level	n = 4

MATERIAL:

Concrete: C30/37

α_{cc} = 0.85

Creep coefficient: φ = 2.3

Steel: fyk=410 [MPa]

LOADS:

Total bending moment:	M = 168	[kNm]
Bending moment from quasi-permanent combination:	M = 137	[kNm]
Compression force:	N = 776	[kNm]

REINFORCEMENT:

5 bars ϕ20 at both sides of the section (*Fig.1.9*)

IMPORTANT STEPS:

Define the geometry of the column and the buckling model in *Buckling length* dialog (*Fig.1.2*). The direction considered is direction Y (the unidirectional bending option will be enabled in next steps).

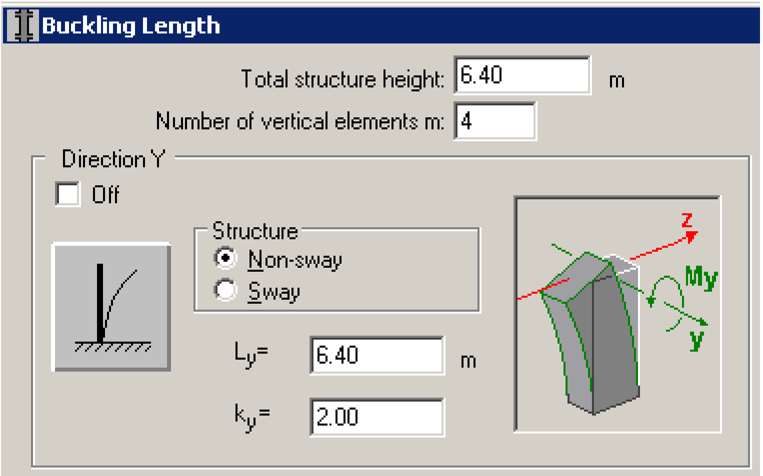


Fig. 1.2 Buckling parameters

Define the loads *(Fig.1.3)* and the parameter M_{0Eqp} / M_{0Ed} (ratio of quasi permanent moment to total moment) – denoted in load table as Nd/N.

No.	Case	Nature	Group	N (kN)	MyA (kN*m)	MyB (kN*m)	MyC (kN*m)	MzA (kN*m)	MzB (kN*m)	MzC (kN*m)	Nd/N	γ
1	DSGN1	design	1	776.00	0.00	168.00	100.80	0.00	0.00	0.00	0.60	1.00

Fig. 1.3 Loads

Set creep coefficient as fixed value in *Story parameters* dialog.
Set proper concrete and steel with fyk=410MPa (34GS) in *Calculation Options*. In order to select steel different than available by default for EN1992-1-1 code (i.e. with fyk=410MPa) which is used in [1], select PN_2002# database in *Job Preferences/Databases/Reinforcing bars (Fig.1.4)*.

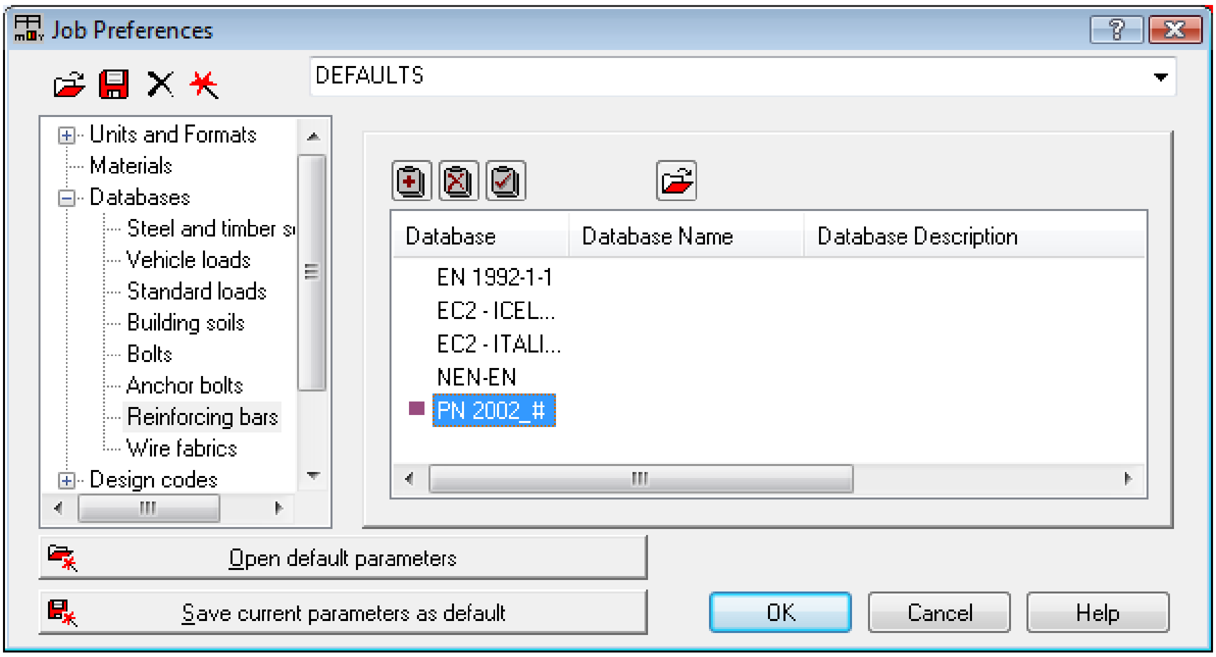


Fig. 1.4 Selection of steel database corresponding to [1]

Select proper second-order analysis method in *Calculation options/General* dialog (*Fig.1.5*).

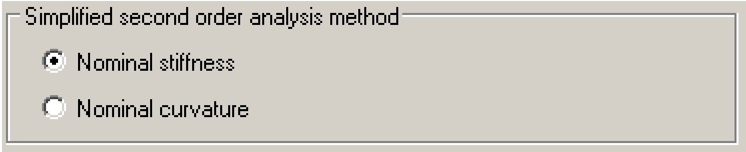


Fig. 1.5 Selection of second order analysis method

In order to enable unidirectional bending analysis, select "Design for simple bending" in *Calculation options/General* dialog (*Fig.1.6*).

Fig. 1.6 Selection of uni-directional bending option

In order to obtain the reinforcement as assumed in [1] select diameter of bars equal to 20mm in *Reinforcement pattern/General* dialog (*Fig.1.7*).

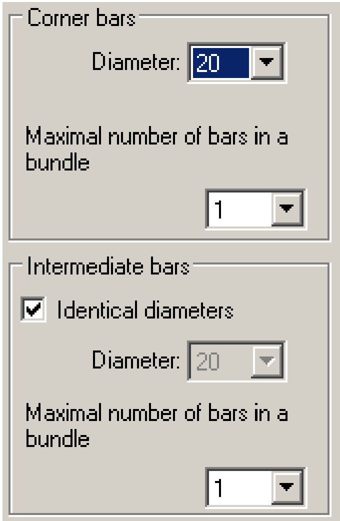


Fig. 1.7 Parameters of reinforcement

The authors of [1] use the partial factor α_{cc} = 0.85. The default value for the general edition of the code is α_{cc} = 1.0. In order to enable the comparison, change the factor to 0.85 in *Job Preferences/Design Codes/Partial factors for a Code EN 1992-1-1:2004 AC:2008/User defined (Fig.1.4).*

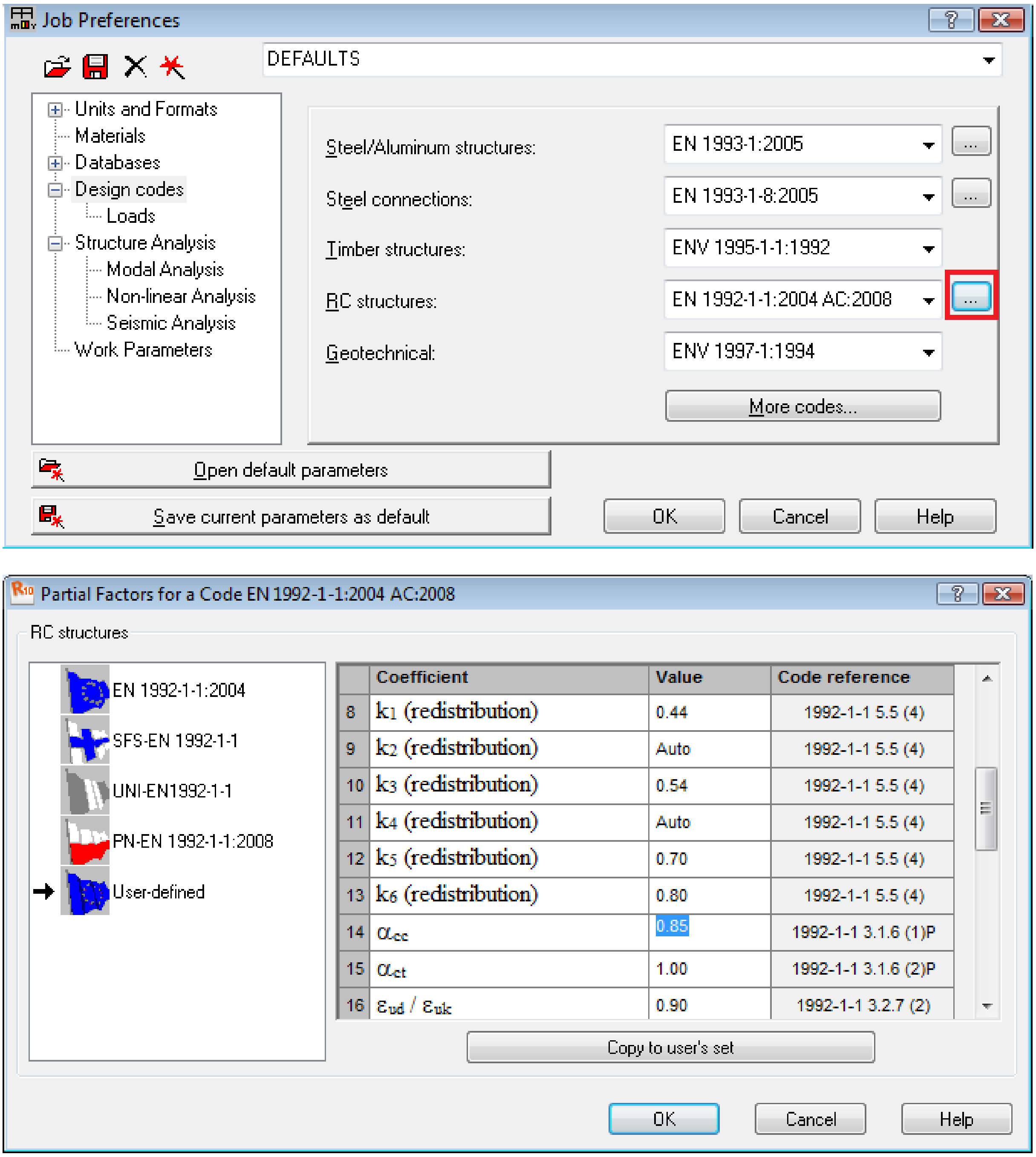


Fig. 1.8 Definition of partial factors

NOTE: The program automatically generates smaller reinforcement (8 ϕ20 for both methods: nominal curvature and nominal stiffness) than assumed in [1] (the capacity is in [1] first verified against the previous edition of Eurocode 2, which gives greater total moment). Since the presented example concerns the comparison of second-order analysis, the reinforcement should be modified to the same form as in [1] (see *Fig.1.9*)

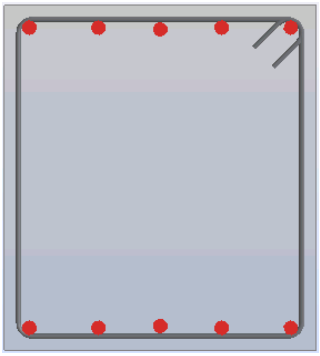


Fig. 1.9 Reinforcement assumed for the calculation (10 ϕ20).

RESULTS OF BUCKLING ANALYSIS - NOMINAL CURVATURE METHOD:

	(Unit)	[1]	Robot (results presented in calculation note)
$\lambda_{\lim}$	(-)	32.2	32.3
α_h	(-)	0.791	0.791
α_m	(-)	0.791	0.791
e_a	(cm)	2.0	2.0
K_r	(-)	1.0	1.0
K_φ	(-)	1.0	1.0
$1/r_0$	(1/m)	0.00863	0.00853*
$1/r_0$	(1/m)	0.00863	0.00853
c	(-)	10	10
e_2	(cm)	13.7 (14.1)**	14.0
M_{Ed}	(kNm)	289.8 (293.9)**	291.97

As can be seen, a very good agreement concerning the final results is obtained, even if some small discrepancies may occur in partial results.

NOTES ON DIFFERENCES IN THE COMPARISON:

* - the difference is due to accuracy of steel strength value used in calculation of $1/r_0$ (the authors of [1] use fixed $f_{yd} = 350MPa$ value, while program uses $f_{yd} = f_{yk}/\gamma_s = 357MPa$

** - the value of e_2 calculated in [1] is erroneous (simple calculation error was apparently made in handbook). The corrected values are presented here in parentheses.

AUTODESK

ANALYSIS OF RESULTS FOR NADs:

In this section, the same example is calculated for different national editions of Eurocode 2. It has been found that the results for all NADs are exactly the same as for general edition of Eurocode 2, except of the EN 1992-1-1 DK NA:2007 code, where the nominal curvature method is not used. The list of the codes, for which the calculation was carried out is presented below:

PN-EN 1992-1-1:2008
UNI-EN 1992-1-1
SFS-EN 1992-1-1
EN 1992-1-1 DK NA:2007
BS EN1992-1-1:2004 NA2005
NS-EN 1992-1-1:2004/NA:2008
NF EN 1992-1-1/NA:2007

RESULTS OF BUCKLING ANALYSIS - NOMINAL STIFFNESS METHOD:

	(Unit)	[1]	Robot (results presented in calculation note)
J_s	(cm^4)	14500	14442
J_c	(cm^4)	785000**	468750
EJ	(kNm^2)	38670**	34285
N_b	(kN)	2330	2065
β	(-)	$\pi^2/12 = 0.8225$	$\pi^2/8 = 1.2337$ ***
M_{Ed}	(kNm)	258.9	319.79***

NOTES ON DIFFERENCES IN THE COMPARISON ABOVE

****** - apparently, the calculation error was made in [1]. The Robot gives proper value of J_c.

******* - the authors of [1] take the value of $c_0 = 12$ for triangular distribution of moment. In Robot program however, this value is by default assumed as $c_0 = 8$ since the exact distribution of moment along the height of the column is not known (thus, more unfavorable case is chosen). Thus, β is taken as $\pi^2/8 = 1.2337$ when the moment in the mid-height (Mc) is not fixed by the user in the load definition dialog and $\beta = 1$ is assumed when Mc is fixed (i.e. when neither 5.8.7.3 (2) nor (3) can be applied). It naturally leads to the greater (in this particular case by 20%), but at the same time safer, value of total moment.

ANALYSIS OF RESULTS FOR NADs:

In this section, the same example is calculated for different national editions of the Eurocode 2. The results of calculation are compared in the table below, along with the values of coefficients which allows you to understand the possible differences for different NADs.

Code	γ_c	α_{cc}	Design moment M_{Ed}
EN 1992-1-1:2004 AC:2008	1.5	1.0	317.38 kN
PN-EN 1992-1-1:2008	1.4	1.0	319.79 kN
UNI-EN 1992-1-1	1.5	0.85	311.39 kN
SFS-EN 1992-1-1	1.5	0.85	311.39 kN
EN 1992-1-1 DK NA:2007	1.45	1.0	318.57 kN
BS EN1992-1-1:2004 NA2005	1.5	0.85	311.39 kN
NS-EN 1992-1-1:2004/NA:2008	1.5	0.85	311.39 kN
NF EN 1992-1-1/NA:2007	1.5	1.0	317.38 kN

As it can be seen, the results may slightly differ for some NADs which is due to the different partial material coefficients for concrete. Due to this, the K_c coefficient, being a function of design strength varies, and thus varies the stiffness *EJ*.

CONCLUSIONS

The results obtained in Robot are in agreement with those obtained in [1] for nominal curvature method. For nominal stiffness method, the discrepancy is found due to the value of coefficient describing moment distribution assumed in Robot. Since the exact distribution of moment along the height of the column is not known in the program, more unfavorable case is chosen, thus greater total moment is calculated by the program. The calculations have also been carried out for different NADs available in Robot and compared against the general edition of the code.

LITERATURE

[1] Bases of designing of reinforced and prestressed concrete structures according to Eurocode 2 (in Polish). Dolnośląskie Wydawnictwa Edukacyjne, Wroclaw 2006

[2] Eurocode 2 EN 1992-1-1:2004 AC:2008

3. Eurocode 2 EN 1992-1-1:2004 AC:2008 - RC slabs (punching)

VERIFICATION PROBLEM 1
- Punching capacity of slab without shear reinforcement

Example based on:
[1] Bases of designing of reinforced and prestressed concrete structures according to Eurocode 2 (in Polish). Dolnośląskie Wydawnictwa Edukacyjne, Wroclaw 2006, Example 9.2.5.1, pp. 486 *

*** -** NOTE: the reference handbook [1] is based on the edition of Eurocode 2 EN1992-1-1:2004. The calculations in program Autodesk Robot Structural Analysis Professional are based on the last edition with the corrections EN1992-1-1:2004 AC:2008. However, the last correction does not introduce any changes within the range of the calculations presented in the examples.

DESCRIPTION OF THE EXAMPLE:

Calculate the punching capacity of the internal node of slab-column structure.

GEOMETRY:

slab thickness:	h=24.0	[cm]
effective depth (average):	d=20.9	[cm]
column section:	30x30	[cm]

REINFORCEMENT:

reinforcement area:	A_x=A_y=16.08	[cm^2/m]
reinforcement ratio:	ρ_x=ρ_y=0.0077	

MATERIAL:

Concrete:	fck = 15	[MPa]

IMPORTANT STEPS:

In the Structure model/Geometry view, define the slab with the supporting column in the middle. The slab should be of proper size, so the column is not located at any of its edges. Define the thickness of the slab in *FE Thickness* dialog (*Fig.1.1*). Set proper concrete type. Since there is no concrete with fck=15MPa in the default Eurocode 2 material database, the new material should be added in the *Job Preferences* dialog. From the left-hand side list, select materials and then use *Modification* button (*Fig.1.2*). On the Concrete Tab set the parameters for new concrete type and use *Add* button. Define new reinforcement pattern in the *Plate and Shell reinforcement type.* On the *Materials* tab, check the option *As in structure model* for concrete. Set proper cover of bars on the *Reinforcement* tab (*Fig.1.3*). Having calculated the structure model and the RC required reinforcement send the slab to provided RC calculations. On the *Slab-provided reinforcement* view, in *Reinforcement pattern/General* dialog select reinforcement with bars (*Fig.1.4*). On the *Bars* tab (*Fig.1.5*), set diameters to 12mm, and the spacing of top bars to 7cm (in order to obtain the reinforcement ratio as in Handbook example). Now, the calculations of real reinforcement, along with punching calculations may be carried out.

AUTODESK

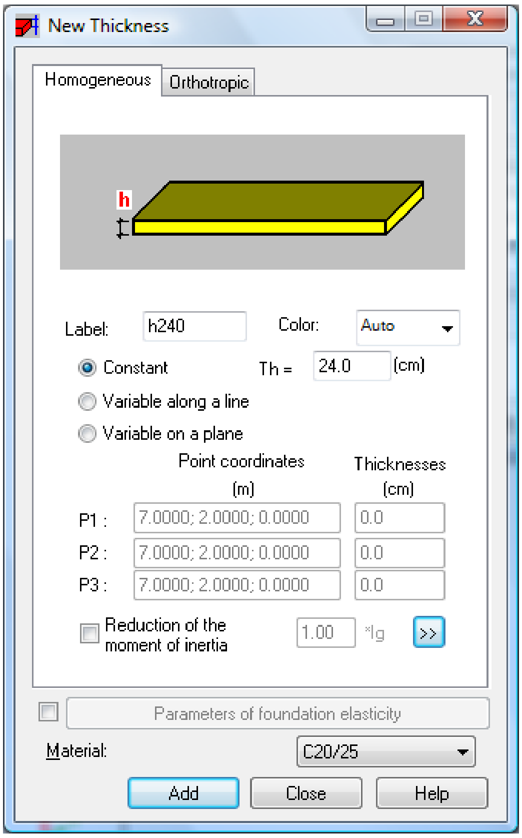


Fig. 1.1 Slab thickness

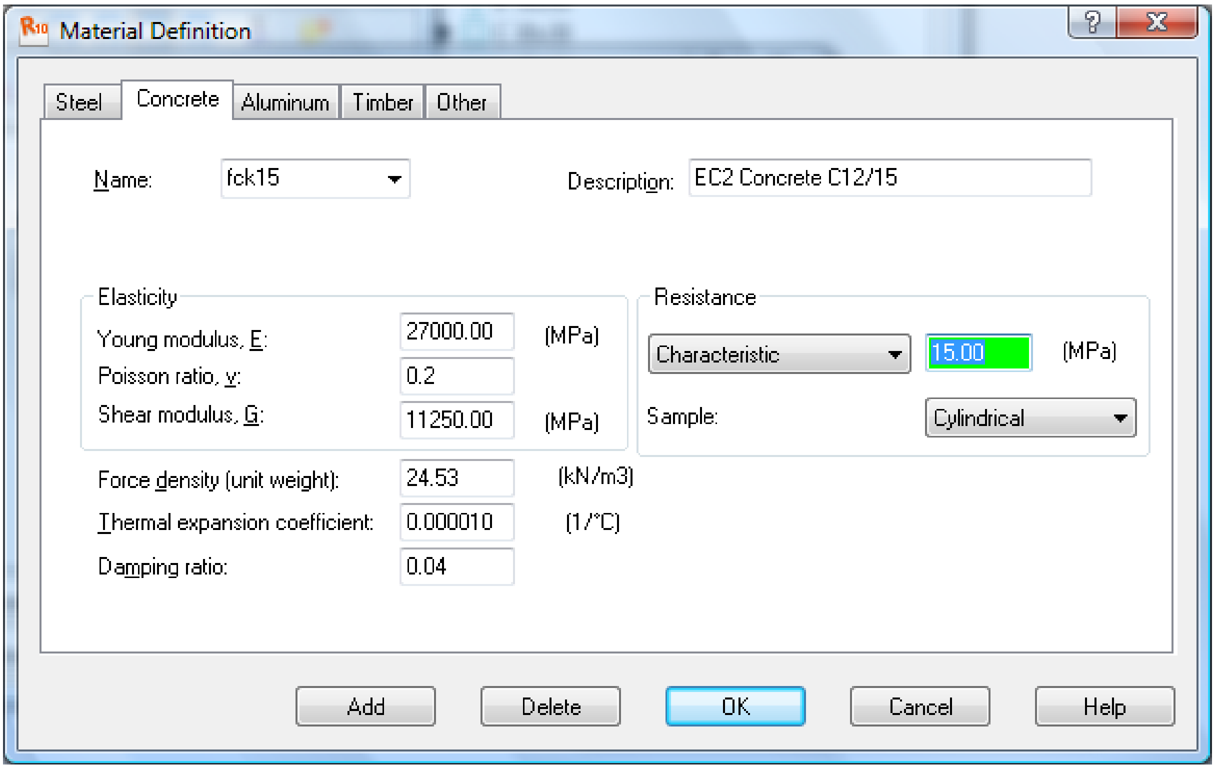


Fig. 1.2 Definition of new concrete type

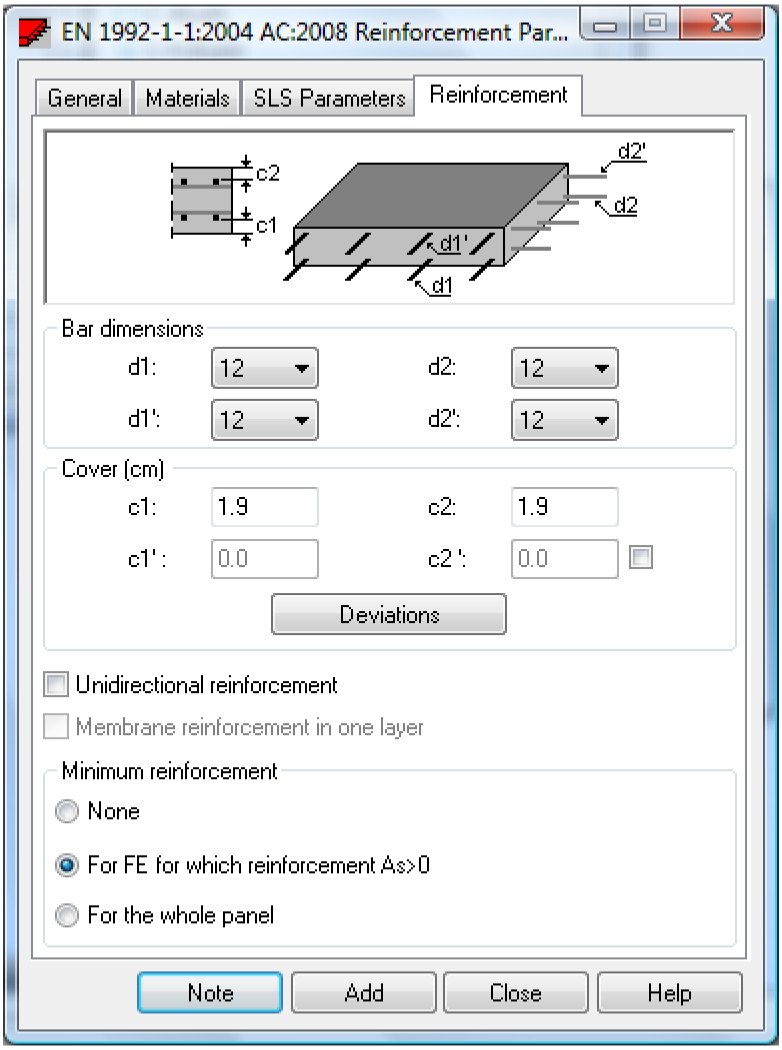


Fig. 1.3 Definition of covers of reinforcement

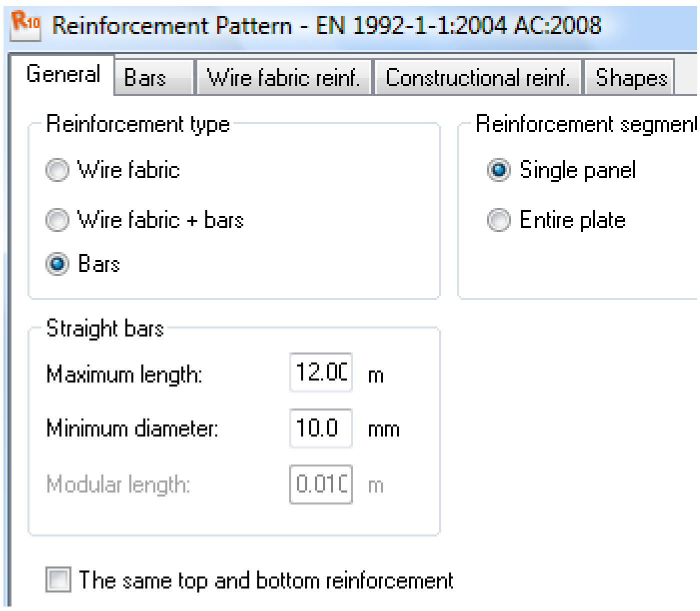


Fig. 1.4 Selection of reinforcement with bars

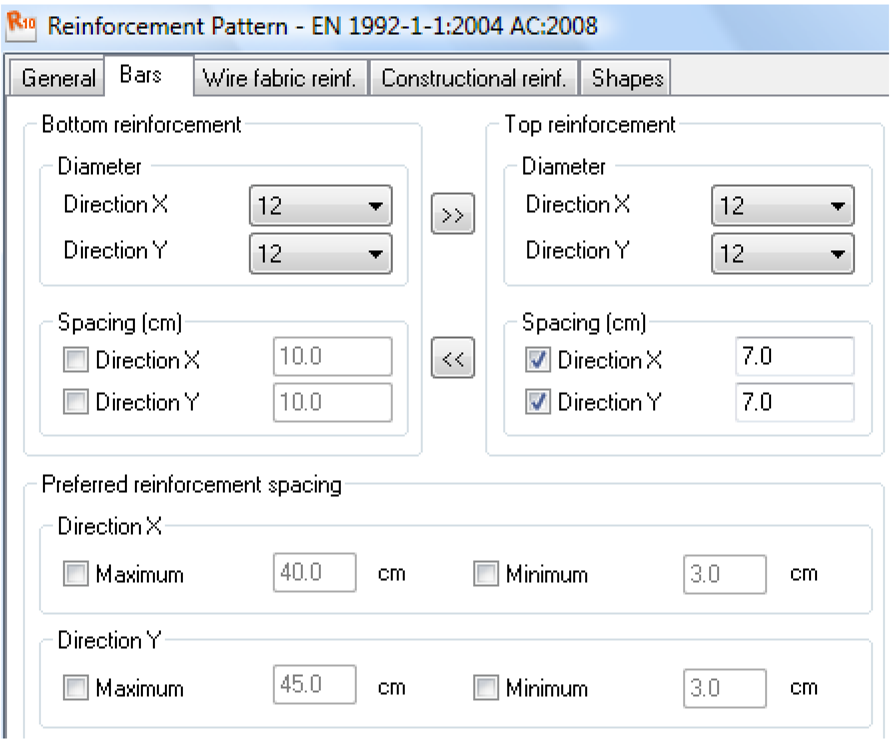


Fig. 1.5 Definition of spacing and diameters of reinforcement

RESULTS OF PUNCHING CALCULATIONS:

The results of punching calculations may be seen on Slab-punching view (*Fig.1.6*). The punching capacity (denoted as Qadm) is compared with Handbook result in the table below.

	[1]	Robot
Punching capacity	429 kN	430 kN

As can be seen, the results of the capacity calculation are in a very good agreement.

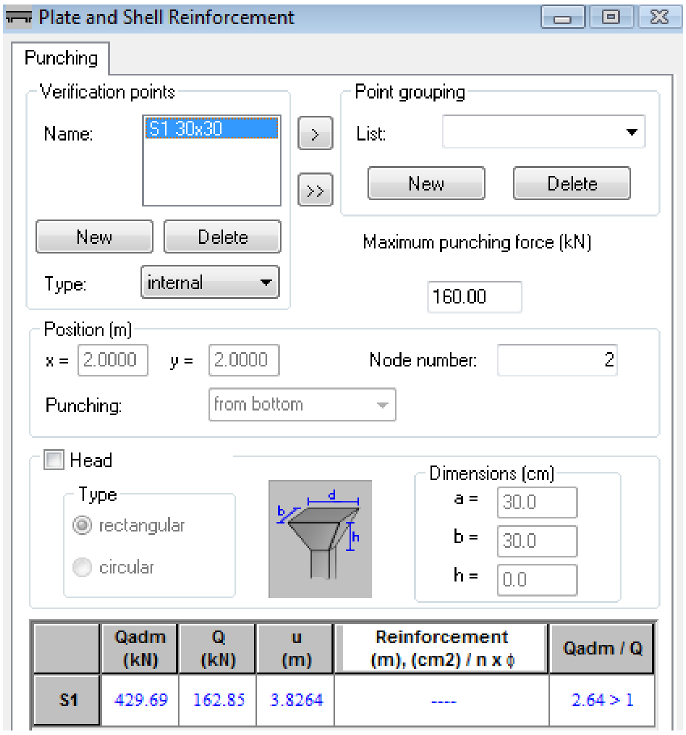


Fig. 1.6. Punching calculations dialog.

ANALYSIS OF RESULTS FOR NADs:

The presented example has been calculated for the general edition of Eurocode 2 [2]. In this section, the same example is calculated for different national editions of the Eurocode 2. The results of calculation are compared in the table below, along with the values of partial coefficients which allows you to understand the possible differences for different NADs.

Code	γ_c	Punching capacity
EN 1992-1-1:2004 AC:2008	1.5	430 kN
PN-EN 1992-1-1:2008	1.5	430 kN
UNI-EN 1992-1-1	1.4	460 kN
EN 1992-1-1 DK NA:2007	1.45	445 kN
BS EN1992-1-1:2004 NA2005	1.5	430 kN
NS-EN 1992-1-1:2004/NA:2008	1.5	430 kN
NF EN 1992-1-1/NA:2007	1.5	457 kN

As it can be seen above, the results may slightly differ for some NADs due to the different material coefficients. However, the manual calculations carried out show that the results are correct for all cases.

VERIFICATION PROBLEM 2 - Punching capacity of slab without shear reinforcement for Finnish NAD

Example based on:
Manual calculation

DESCRIPTION OF THE EXAMPLE:

Based on Finnish NAD SFS-EN 1992-1-1 [3], calculate the punching capacity of the internal node of slab-column structure without punching reinforcement. In this example, the same data as in Verification problem 1 is assumed, except for the concrete type, which is taken as C20/25 here.

GEOMETRY:

slab thickness:	h=24.0	[cm]
effective depth (average):	d=20.9	[cm]
column section:	30x30	[cm]

REINFORCEMENT:

reinforcement area:	A_x=A_y=16.08	[cm^2]
reinforcement ratio:	ρ_x=ρ_y=0.0077	

MATERIAL:

Concrete: C20/25

FORCES IN THE NODE:

Vertical force: N = 192 kN

Moments: M_x = 24 kN

M_y = 40 kN

CALCULATION OF PUNCHING CAPACITY:

$$V_c = k\beta(1+50\rho)udf_{ctd} = 210kN \qquad (2.38)$$

$$k = 1.6 - d[m] = 1.391 \qquad (\rho c = 2500\ kg/m^3)$$

$$d = 0.209m$$

$$\rho = 0.0077$$

$$u = 2\left(c_x + d + c_y + d\right) = 2.036m$$

$$c_x = c_y = 0.3m$$

$$f_{ctd} = f_{ctk} / \gamma_c = 1.0MPa$$

$$f_{ctk} = 1.5MPa$$

$$\gamma_c = 1.5$$

$$\beta = \frac{0.40}{\left(1 + 1.5\frac{e}{\sqrt{A_u}}\right)} = 0.256$$

$$e = \sqrt{e_x{}^2 + e_y{}^2} = 0.243m$$

$$e_x = M_y / N = 0.125m$$

$$e_y = M_x / N = 0.208m$$

$$A_u = 0.426m^2$$

The results of punching calculations may be seen on Slab-punching view (*Fig.2.1*). The value of $V_{Rd,c}$ calculated by the program (denoted as Q in Punching dialog) is in very good agreement with the one calculated above (see table below).

	Manual calculation	Robot
Punching capacity	211 kN	211 kN

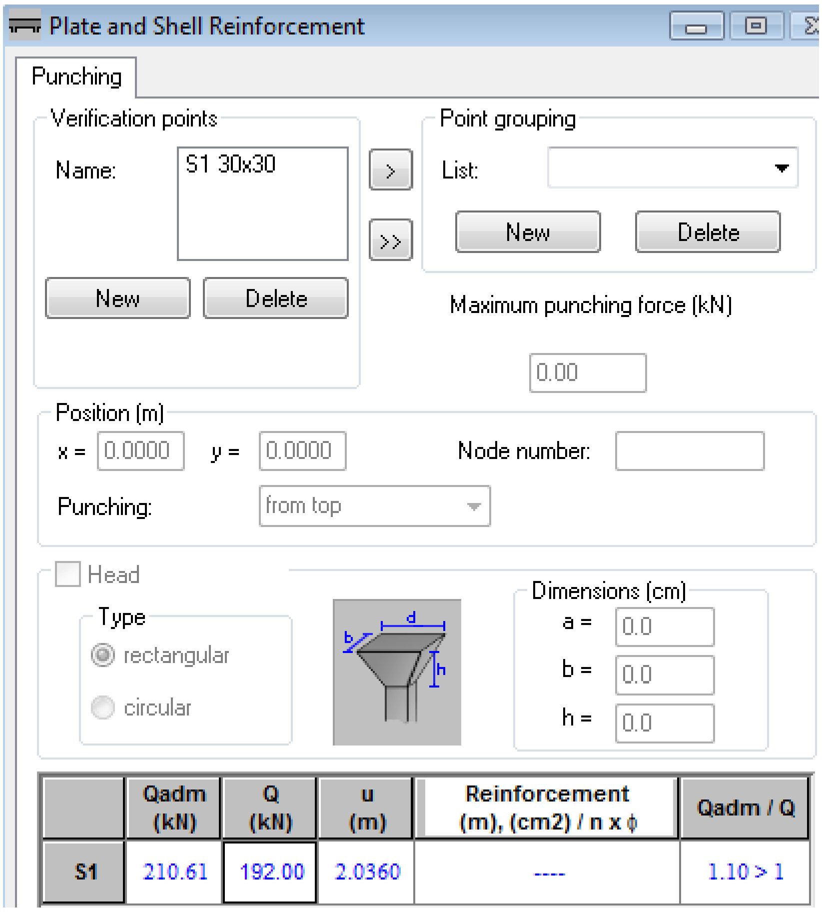


Fig. 2.1. Punching calculations dialog.

VERIFICATION PROBLEM 3 - Calculation of punching force for eccentrically applied support reaction

Example based on:
Manual calculation

DESCRIPTION OF THE EXAMPLE:

Based on general edition of Eurocode 2 [2], calculate the tangent stress and punching force in the internal node of slab-column structure with eccentrically applied load. In this example, the results of the Robot program are compared against the manual calculation.

GEOMETRY:

slab thickness:	h=24.0	[cm]
effective depth (average):	d=20.9	[cm]
column section:	c_x=50	[cm]
	c_y=30	[cm]

REINFORCEMENT:

reinforcement area:	A_x=A_y=16.08	[cm^2]
reinforcement ratio:	ρ_x=ρ_y=0.0077	

MATERIAL:

Concrete: C20/25

FORCES IN THE NODE:

Vertical reaction:	V = 192 kN
Moments:	M_x = 24 kN
	M_y = 40 kN

CALCULATION OF β COEFFICIENT:

In Robot, β coefficient is calculated for both directions according to the equation (6.38) [2] modified for biaxial bending into a form:

$$\beta = 1 + k_x \frac{M_x}{V}\frac{u}{W_x} + k_y \frac{M_y}{V}\frac{u}{W_y} = 1.64$$

$$u = 4.2264m$$

$$k_x = 0.48 \qquad \text{for } \frac{c_y}{c_x} = 0.60$$

$$k_y = 0.67 \qquad \text{for } \frac{c_x}{c_y} = 1.67$$

$$W_x = 0.5c_y{}^2 + c_y c_x + 4c_x d + 16d^2 + 2\pi d c_y = 1.706$$

$$W_y = 0.5c_x{}^2 + c_x c_y + 4c_y d + 16d^2 + 2\pi d c_x = 1.881$$

$$\nu_{Ed} = \beta \frac{V_{Ed}}{ud} = 387kPa$$

$$Q = \nu_{Ed} \cdot A_u = 342kN$$

$$A_u = ud = 0.883m^2$$

The results of punching calculations may be seen on Slab-punching view (*Fig.3.1*). The value of punching force calculated by the program (denoted as Q in Punching dialog) is in very good agreement with the one calculated above (see table below).

	Manual calculation	Robot
Punching force	342 kN	345 kN

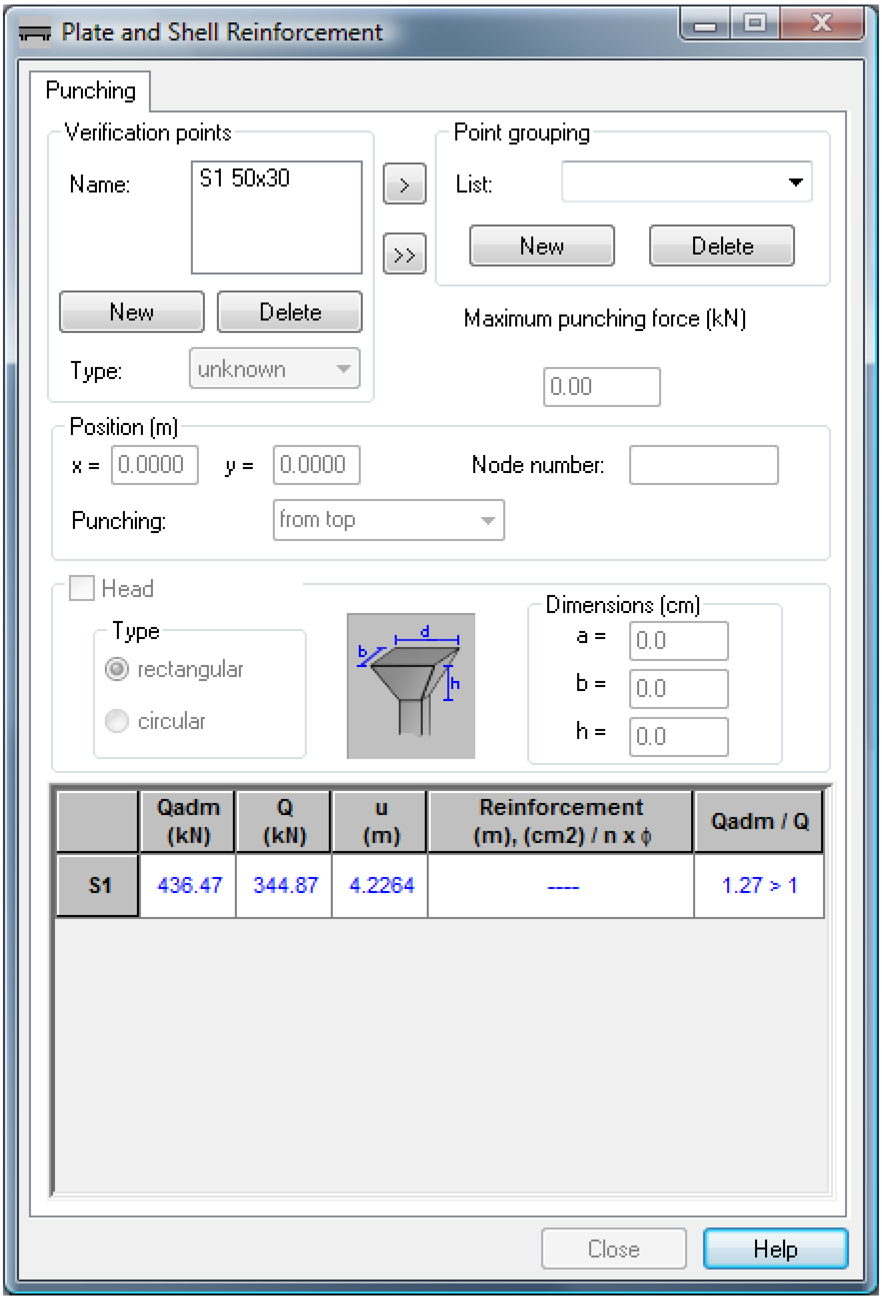


Fig. 3.1. Punching calculations dialog.

ANALYSIS OF RESULTS FOR NADs:

The presented example has been calculated for the general edition of Eurocode 2 [2]. In this section, the same example is calculated for different national editions of Eurocode 2. The results of calculation are compared in the table below.

Code	Punching capacity
EN 1992-1-1:2004 AC:2008	345 kN
PN-EN 1992-1-1:2008	345 kN
UNI-EN 1992-1-1	345 kN
EN 1992-1-1 DK NA:2007	345 kN
BS EN1992-1-1:2004 NA2005	345 kN
NS-EN 1992-1-1:2004/NA:2008	345 kN
NF EN 1992-1-1/NA:2007	345 kN

As it can be seen, the results for different NADs are equal.

VERIFICATION PROBLEM 4
- Punching capacity of slab with shear reinforcement

Example based on:
[1] Bases of designing of reinforced and prestressed concrete structures according to Eurocode 2 (in Polish). Dolnośląskie Wydawnictwa Edukacyjne, Wroclaw 2006, Example 9.6.1, pp. 501 *

*** -** NOTE: the reference handbook [1] is based on the edition of Eurocode 2 EN1992-1-1:2004. The calculations in program Autodesk Robot Structural Analysis Professional are based on the last edition with the corrections EN1992-1-1:2004 AC:2008. However, the last correction does not introduce any changes within the range of the calculations presented in the examples.

DESCRIPTION OF THE EXAMPLE:

Calculate the punching reinforcement for the internal node of slab-column structure.

GEOMETRY:

slab thickness:	h=24.0 [cm]
spacing of columns:	l_x = 6.60 [m]
	l_y = 6.00 [m]
slab thickness:	h=24.0 [cm]
effective depth (average):	d=21.0 [cm]
column section:	40x40 [cm]

REINFORCEMENT:

reinforcement ratio: $\rho_x=\rho_y=0.009$

MATERIAL:

Concrete:	f_{ck} = 20 [MPa]
Steel:	f_{yk} = 355 [MPa] (18G2 steel)

LOADS:

dead loads:	7.5 kN/m^2
live loads:	3.0 kN/m^2
dead load coefficient:	1.35
live load coefficient:	1.50

IMPORTANT STEPS:

In the Structure model/Geometry view define the slab with the supporting column in the middle. The dimensions of the slab should be 6.60x6.00 m. Set the material to C20/25 concrete. Define the thickness of the slab in *FE Thickness* dialog (*Fig.4.1*). In order to select steel different than available by default for EN1992-1-1 code (i.e. with fyk=355MPa) which is used in [1], select PN_2002# database in *Job Preferences/Databases/Reinforcing bars (Fig.4.2)*. Define new reinforcement pattern in the *Plate and Shell reinforcement type.* On the *Materials* tab, check the option *As in structure model* for concrete. Set proper cover of bars on the *Reinforcement* tab (*Fig.4.3*). Define the loads and create manual combination with proper load coefficients.

NOTE:

In the Handbook example [1], there is no detailed calculation of β coefficient. Instead, the simplified rule (Fig. 6.21N from Eurocode 2 [2]) is used and β=1.15 is assumed. Robot calculations of punching stress are based on calculation of β from equation (6.39), [2]. Thus, in the presented example, the loads as defined cause no bending moments at the support, hence β=1.00. In order to enable the comparison of the reinforcement calculations, the punching force in Robot should be as in the reference example [1]. For this purpose, define the additional linear moment of 7.5 kNm/m along the 6m-long edge of the slab. Now, based on the algorithm as presented in verification problem 3, the β coefficient will be equal to that in Handbook [1].

Having calculated the structure model and the RC required reinforcement, send the slab to provided RC calculations. On the *Slab-provided reinforcement* view, in *Reinforcement pattern/General* dialog select reinforcement with bars. On the *Bars* tab (*Fig.4.4*), set diameters to 12mm, and the spacing of top bars to 7cm (in order to obtain the reinforcement ratio as in Handbook example). Now, the calculations of real reinforcement, along with punching calculations may be carried out.

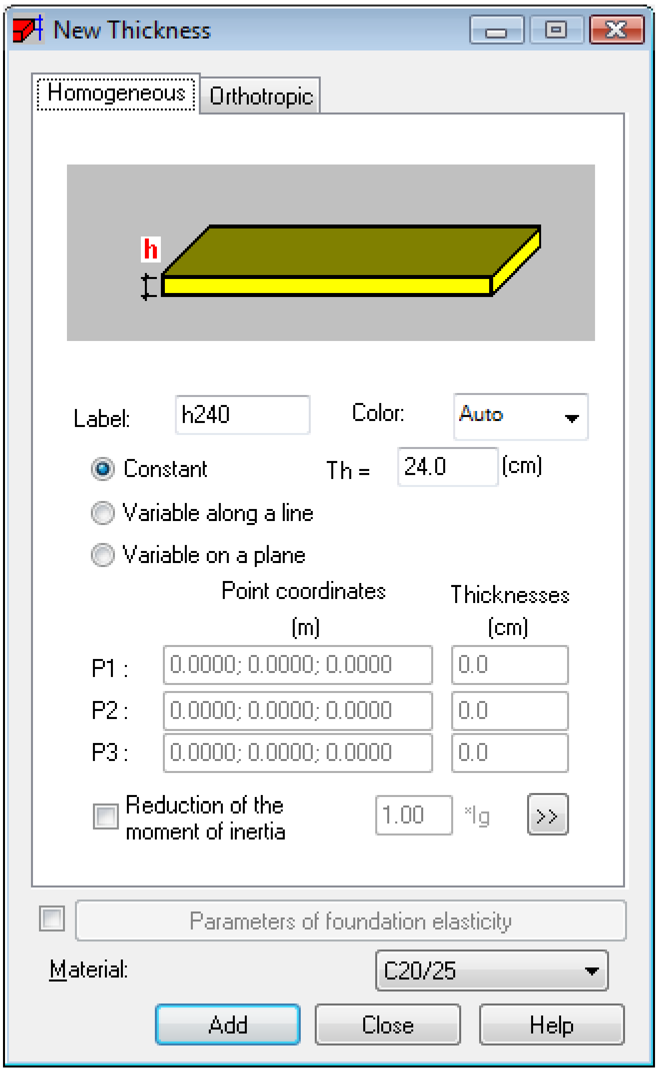


Fig. 4.1. Slab thickness dialog

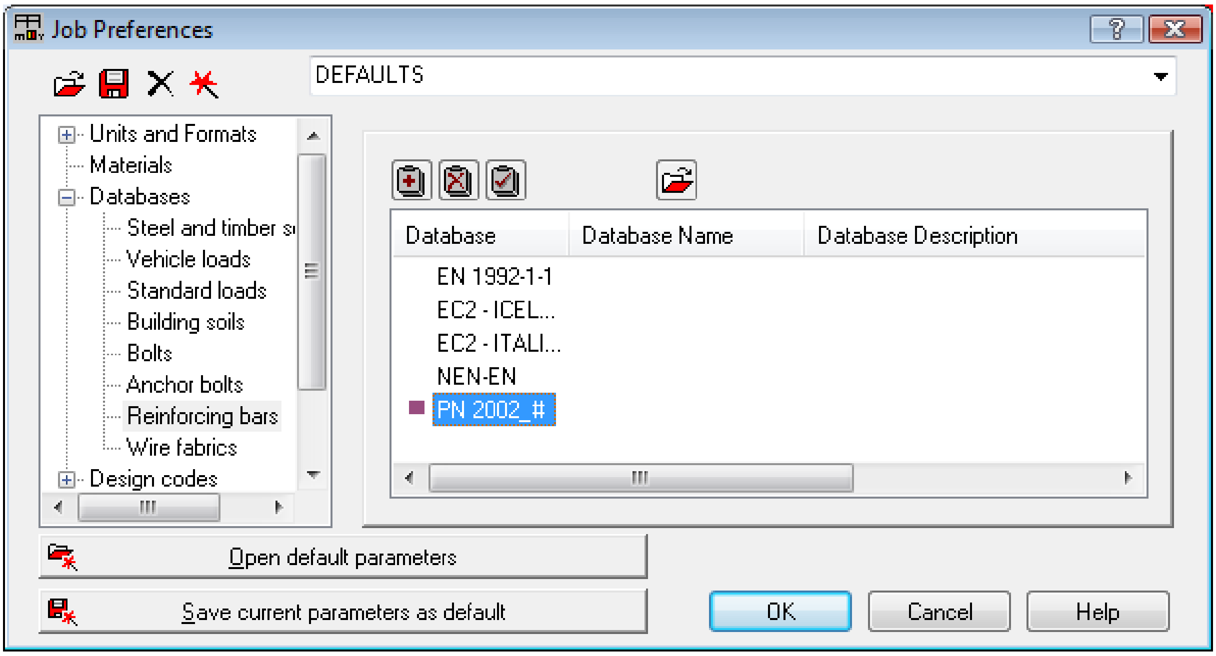


Fig. 4.2 Selection of steel database corresponding to [1]

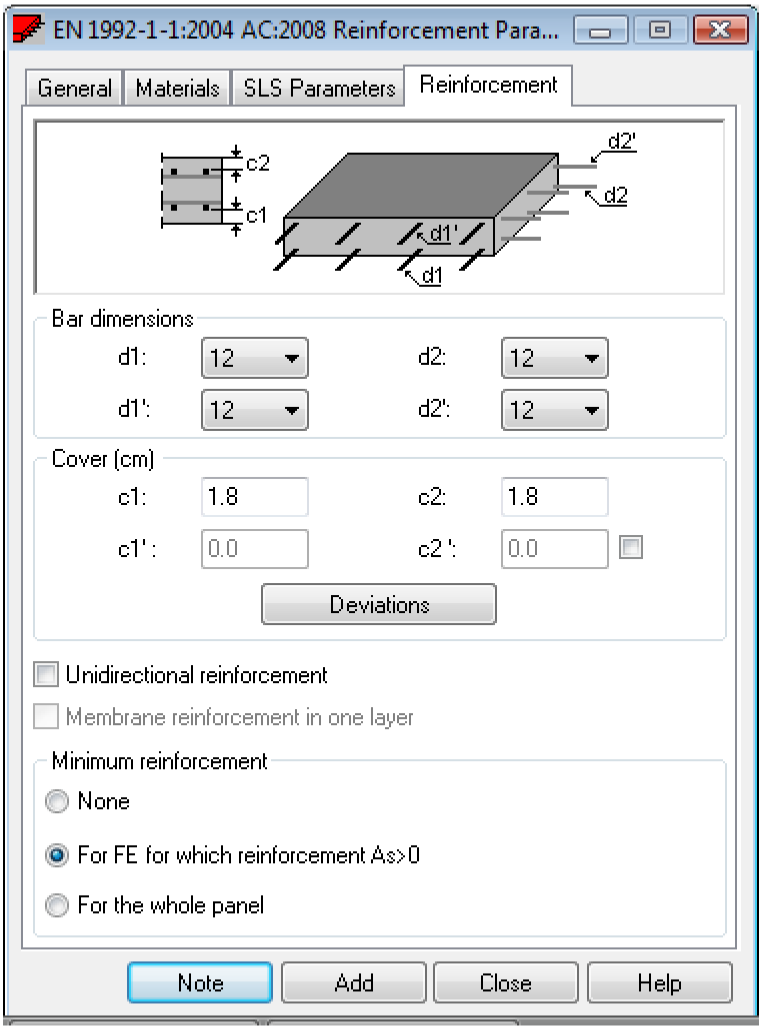


Fig. 4.3 Definition of covers of reinforcement

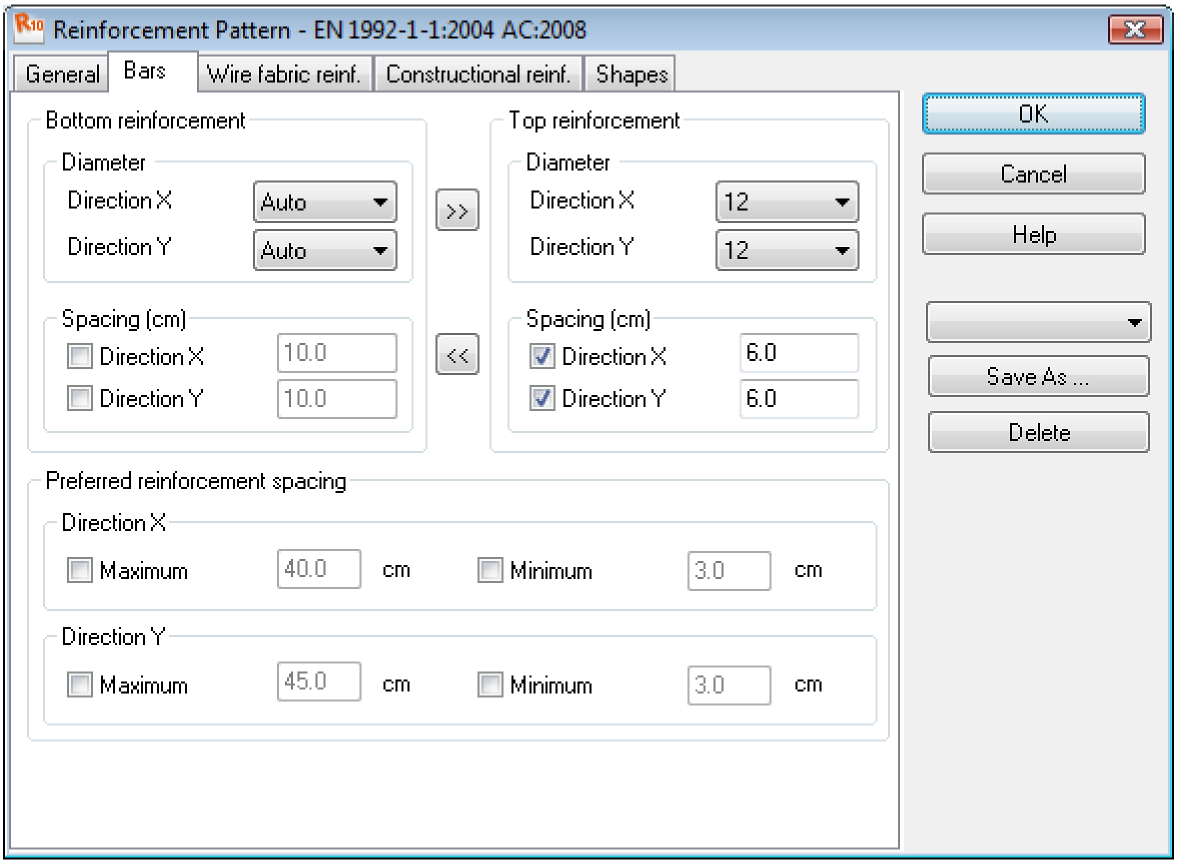


Fig. 4.4 Definition of spacing and diameters of reinforcement

The results of punching calculations may be seen on Slab-punching view (*Fig.4.5*). The value of punching force calculated by the program (denoted as Q in Punching dialog) is in very good agreement with the one calculated above (see table below).

	[1]	Robot
Punching force	666 kN	665 kN

The area of reinforcement in one circumference calculated in [1] was 3.96 cm^2, while in Robot it is 4.14 cm^2 (see table below). This relatively small difference results from the assumed spacing of perimeters assumed during calculation of theoretical reinforcement. In Robot, the spacing is assumed as equal to the maximum allowable value s_r=0.75d, while in [1], the assumed value is smaller than this maximum.

	[1]	Robot
Punching reinforcement	2 perimeters A=3.96 cm^2	2 perimeters A=4.14 cm^2

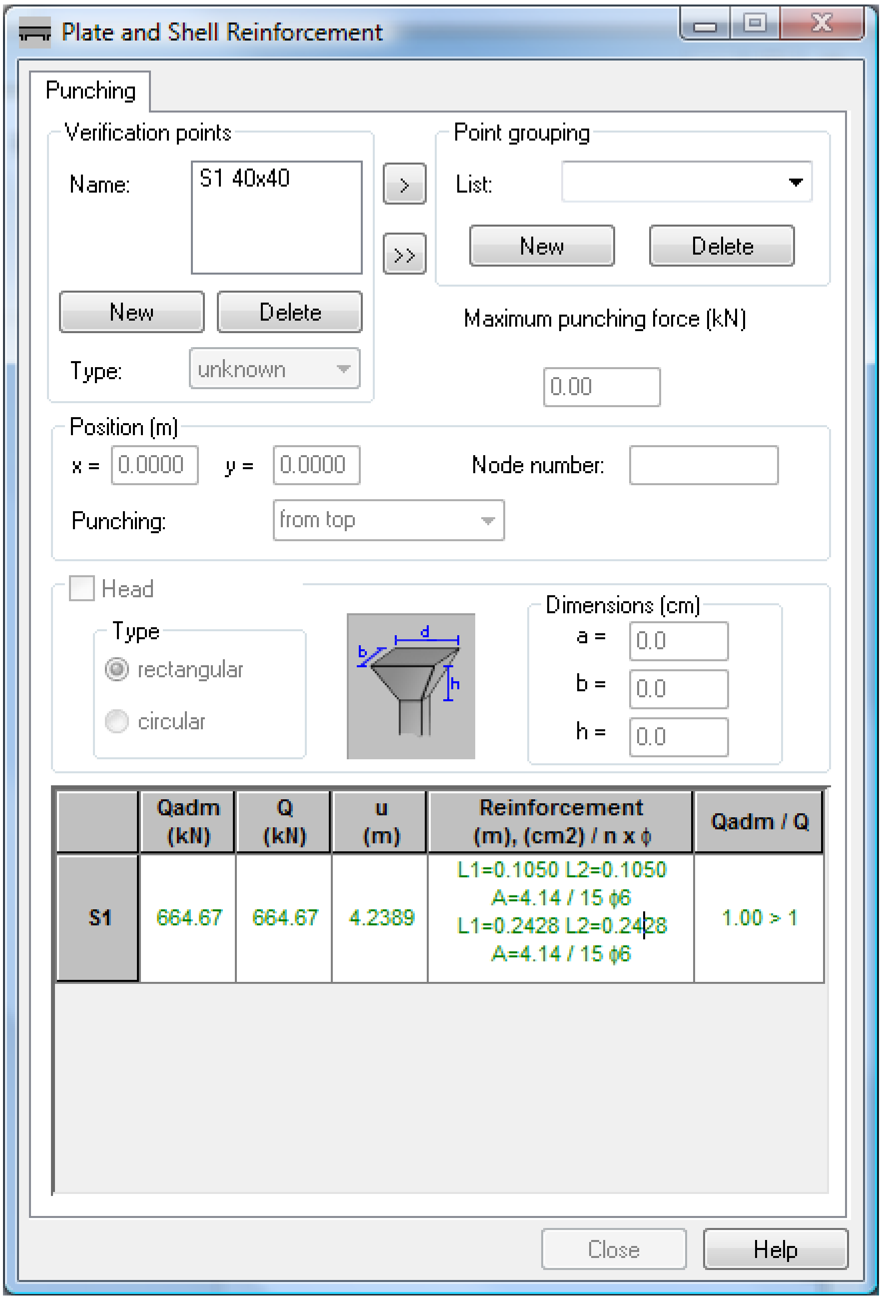


Fig. 4.5. Punching calculations dialog.

As it can be seen in Fig. 4.5, the first perimeter is placed in the distance of 0.105 m from the face of the column, which satisfies the requirement 0.5d.

ANALYSIS OF RESULTS FOR NADs:

The presented example has been calculated for the general edition of Eurocode 2 [2]. In this section, the same example is calculated for different national editions of Eurocode 2. The results of calculation are compared in the table below.

Code	Punching reinforcement
EN 1992-1-1:2004 AC:2008	2 perimeters A=4.14 cm^2
PN-EN 1992-1-1:2008	2 perimeters A=4.14 cm^2
UNI-EN 1992-1-1	2 perimeters A=4.14 cm^2
EN 1992-1-1 DK NA:2007	2 perimeters A=3.99 cm^2
BS EN1992-1-1:2004 NA2005	2 perimeters A=4.14 cm^2
NS-EN 1992-1-1:2004/NA:2008	3 perimeters A=4.14 cm^2
NF EN 1992-1-1/NA:2007	2 perimeters A=3.72 cm^2

As it can be seen, the results may slightly differ for some NADs. The difference concerning the area of reinforcement in one perimeter is a result of different values of material coefficients. The difference concerning the number of perimeters of reinforcement for NS-EN 1992-1-1:2004/NA:2008 is a result of different value of *k* coefficient (6.4.5 (4) [2]), which determines the location of the most external perimeter of the reinforcement. However, the manual calculations carried out show that all these results are correct.

LITERATURE

[1] Bases of designing of reinforced and prestressed concrete structures according to Eurocode 2 (in Polish). Dolnośląskie Wydawnictwa Edukacyjne, Wroclaw 2006.

[2] Eurocode 2 EN 1992-1-1:2004 AC:2008.

[3] National Annex to Eurocode 2 SFS-EN 1992-1-1.

TIMBER

1. Eurocode 5: Design of timber structures

Part 1-1: General - Common rules and rules for buildings

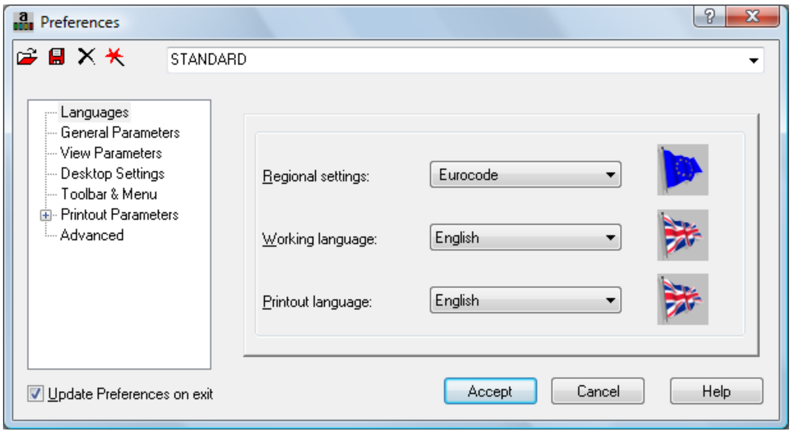
EN 1995-1:2004/A1:2008, March, 2005

GENERAL REMARKS

If you make first step in Robot program you should select preferences corresponding to your example using "Preferences…" or "Job Preferences…" (click Tools).

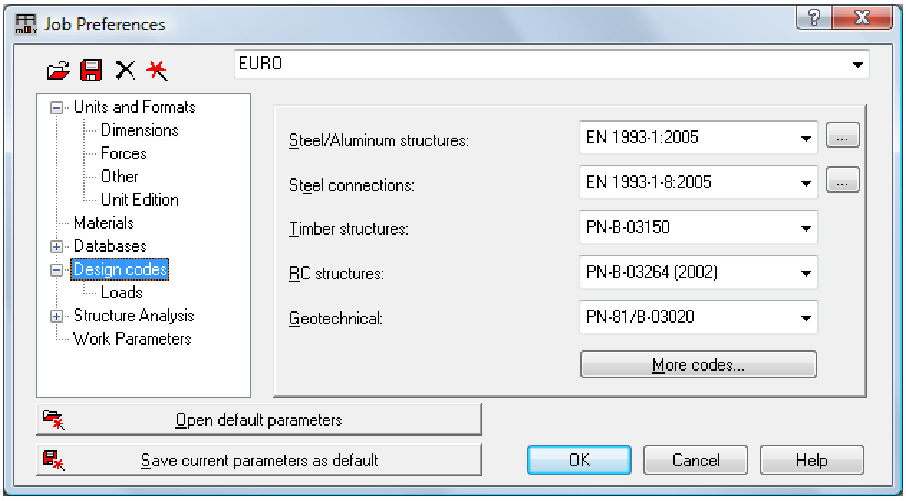
A. Preferences

To specify your regional preferences in PREFERENCES dialog click Tools/ Preferences.
Default PREFERENCES dialog opens e.g.:



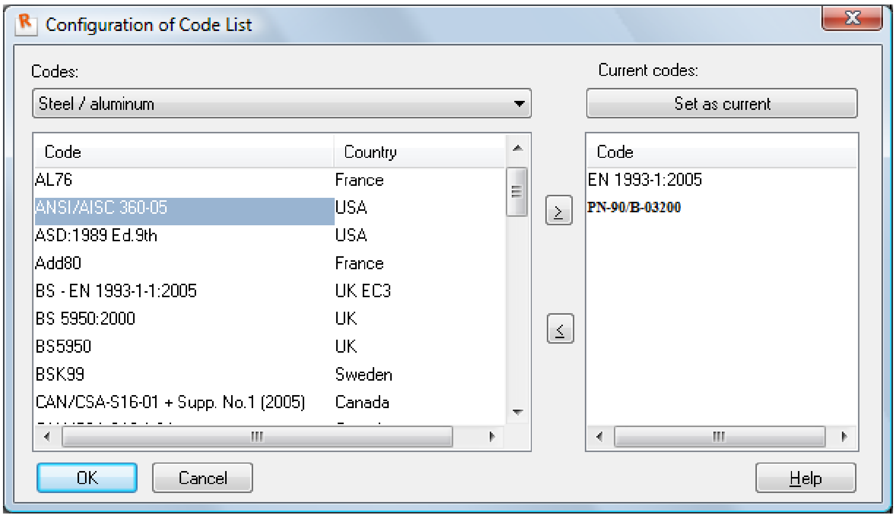
B. Job Preferences

To specify your job preferences in JOB PREFERENCES dialog click Tools/ Job Preferences.
Default JOB PREFERENCES dialog opens, e.g.:

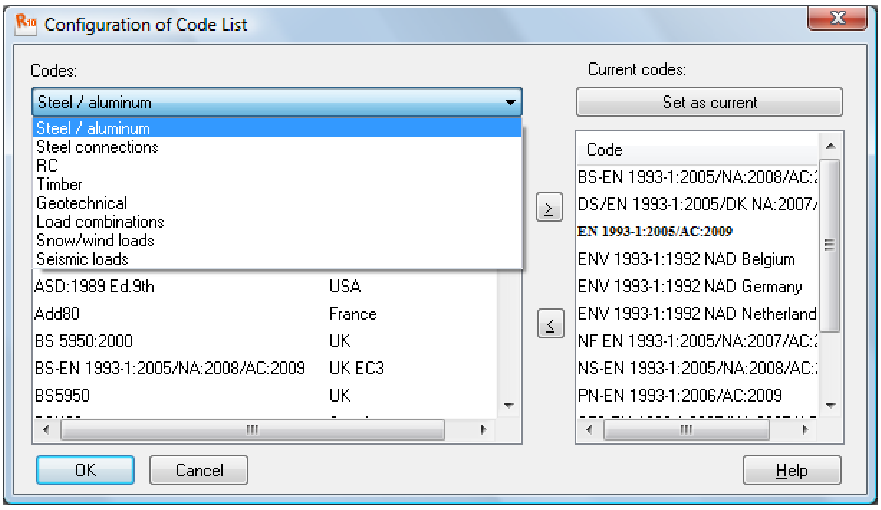


You can define a new type of Job Preferences to make it easier in the future.
First of all, make selection of documents and parameters appropriate for the project conditions from the list view tabs in JOB PREFERENCES dialog.
For example, to choose code, click *Design codes* tab from the left list view; then select code from *Timber structures* selection list or press *More codes* button which opens *Configuration of Code List:*

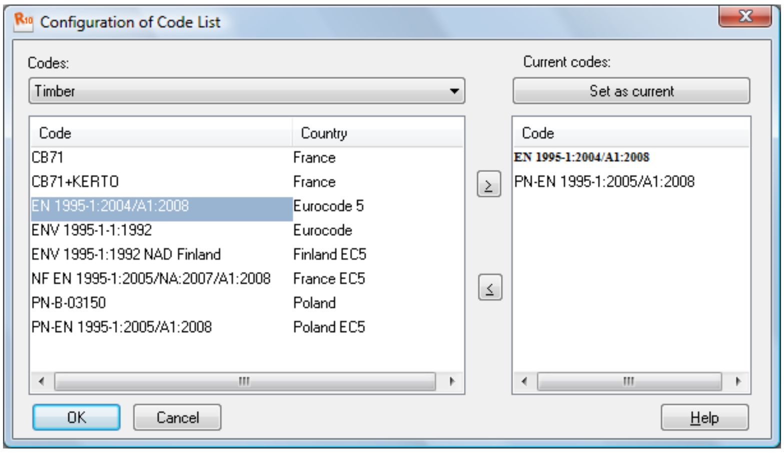
AUTODESK



Select appropriate code category (e.g. *Timber)* from the selection list



A new suitable list view appears. Set code as the *current* code. Press OK.



After the job preferences decisions are set, you can save it under a new name by pressing *Save Job Preferences* icon in the JOB PREFERENCES dialog.

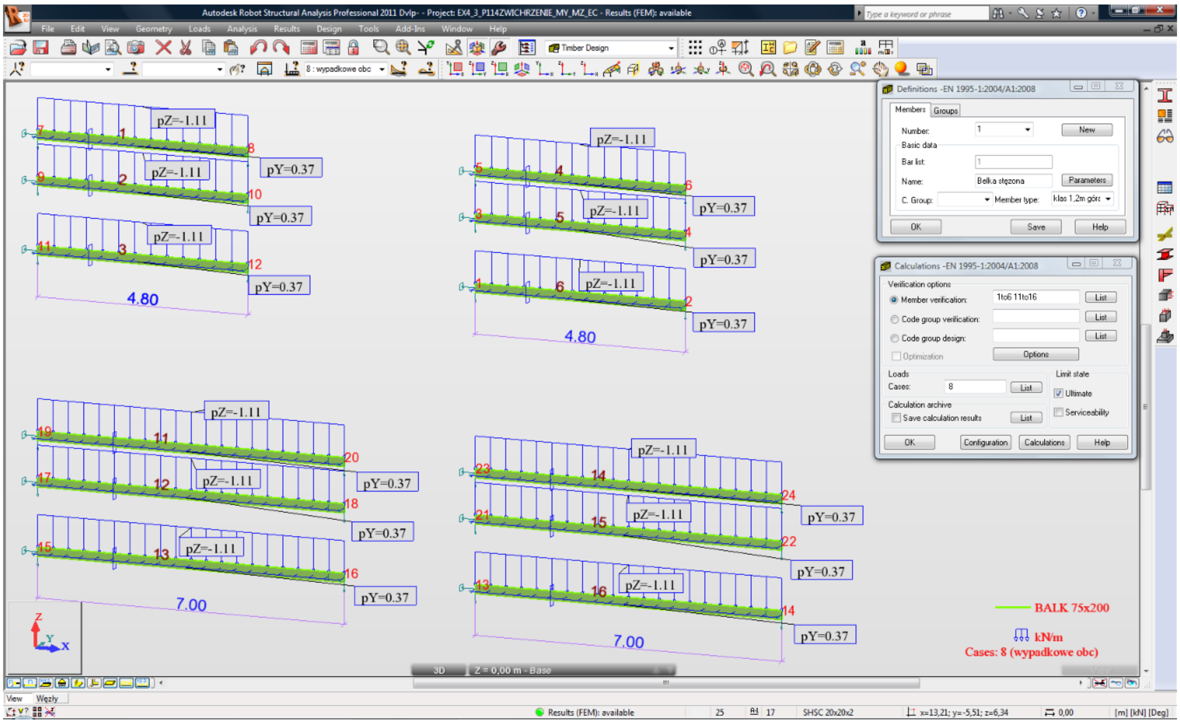
VERIFICATION PROBLEM 1
bending about two main axes with lateral buckling

Example based on “Practical design of timber structures to Eurocode 5”
Hans Larsen and Vahik Enjily
File: EX_4_3p114_bending_My_Mz.rtd

TITLE:
Example 4.3 Solid Timber - Bending About Two Main Axes Restrained or Not Against Torsion
Eurocode5 - EN 1995-1-1:2004

SPECIFICATION:

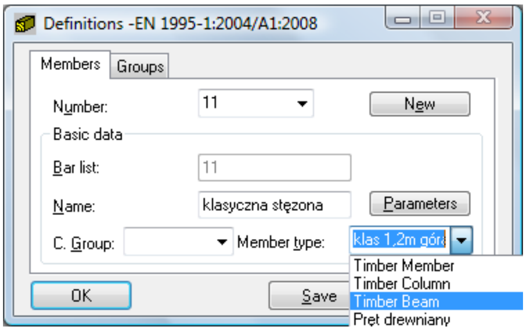
Verify the strength of the C16 cross-section 75×200 mm beams with simply supported spans of 4,8 m and 7,0 m. The beams nº 1, 4, 11, 14 are restrained at 1,2m against torsion. For load case nº 8 loads are assumed as a short-term load and are acting on the bottom (for el. nº 3, 6,13,16) or on the top of the beams (for the others elements) and are equal for all elements: py = 0,37 kN/m, pz = -1,11 kN/m.



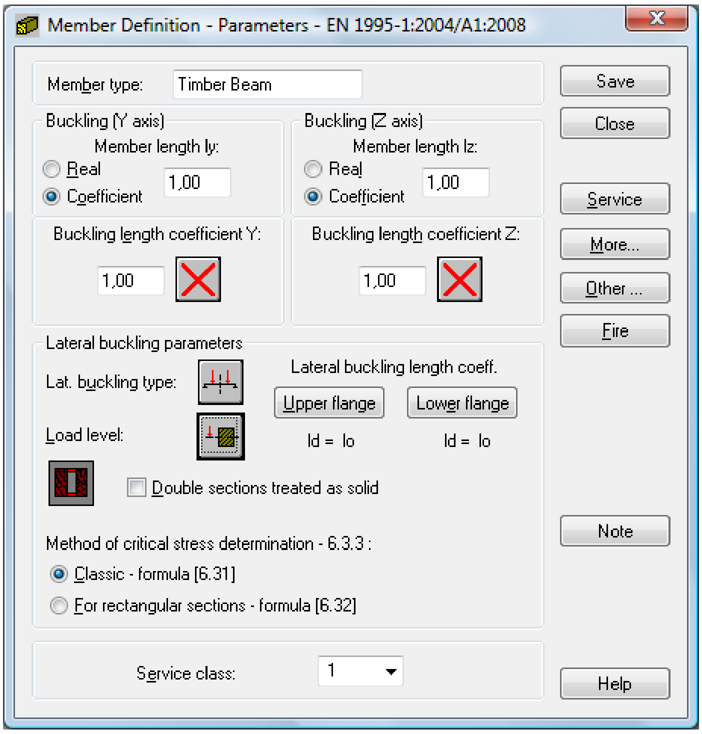
SOLUTION:

After having defined and calculated the structure models, go to [Timber Design] tab.
Define new types of members in accordance with the structure definition in DEFINITIONS dialog.
It can be set in *Member type* selection list.
In this example, the beams numbered 1, 4, 11, 14 are laterally braced at upper flange.

For easier start, the pre-defined type of member (e.g. "*timber beam*") may be initially opened.

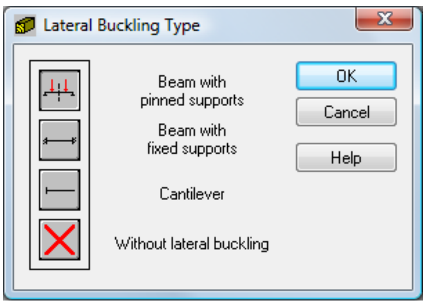


For the selected "Timber Beam" from member type, press the *Parameters* button on *Members* tab. It opens MEMBER DEFINITION - PARAMETERS dialog.

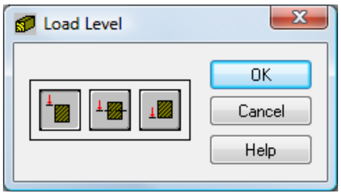


Type a new name in *Member type* editable field. Next, change the parameters to meet the initial data requirements of the structure. Set the following lateral-buckling parameters:

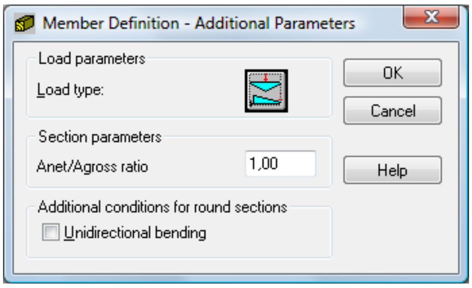
- switch on the appropriate *Lateral buckling type* icon;



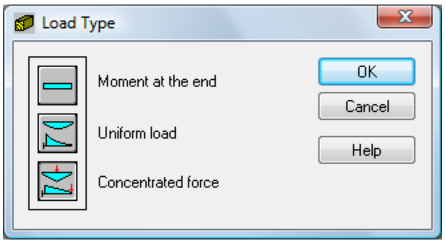
- select the appropriate *Load level* icon



- define the appropriate load type by pressing [More…] button;
  it opens ADDITIONAL PARAMETERS dialog



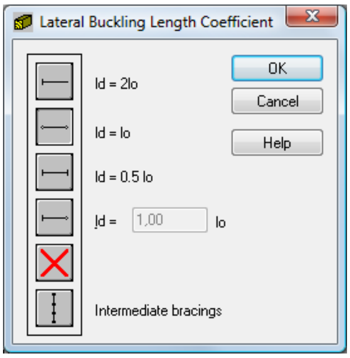
next, choose the load type by pressing the icon - it opens a new dialog:



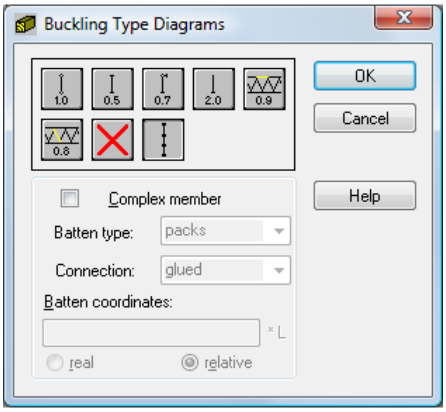
- select *Method of critical stress determination*
- choose *Service class*
- define bracings for *Lateral buckling* and B*uckling:*

→ to define *Lateral buckling length coefficient* for a member, press *Upper/Lower flange* button or the buckling type icon in [MEMBER DEFINITION-MEMBER] dialog

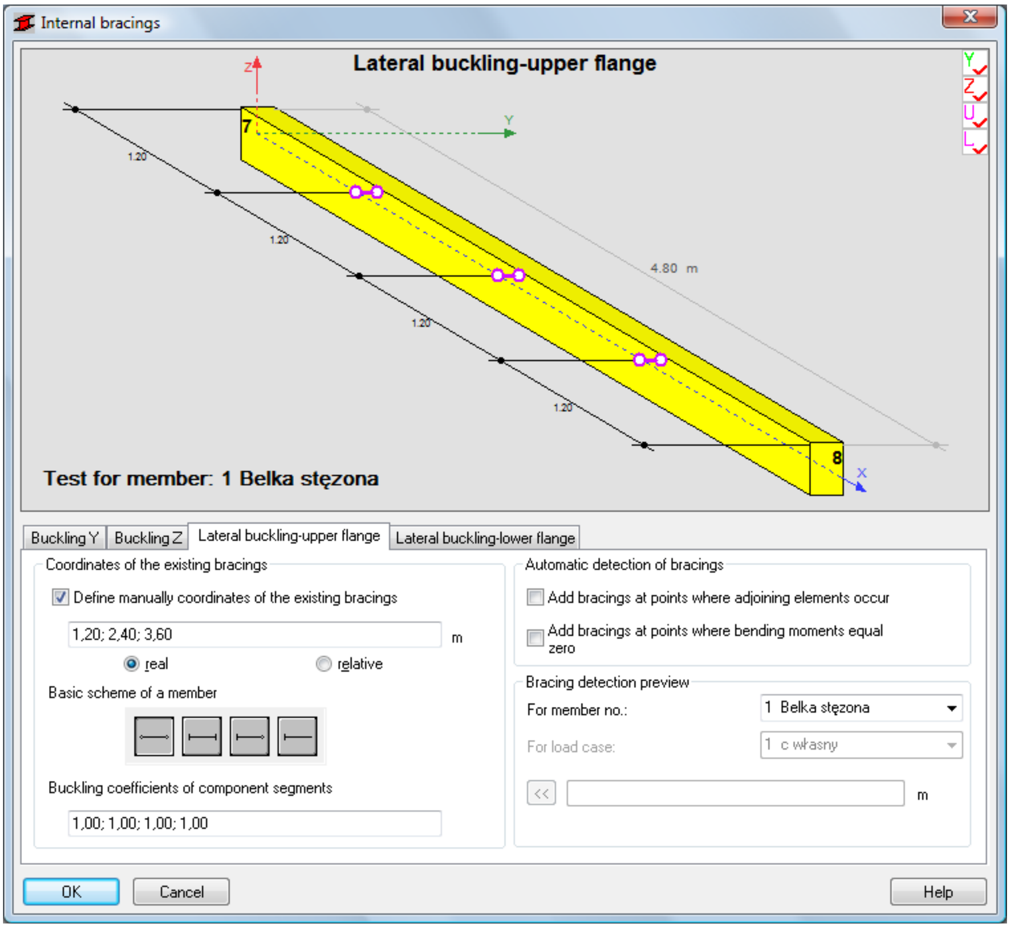
The first method opens LATERAL BUCKLING LENGTH COEFFICIENTS dialog:



The second one opens BUCKING TYPE DIAGRAMS dialog:

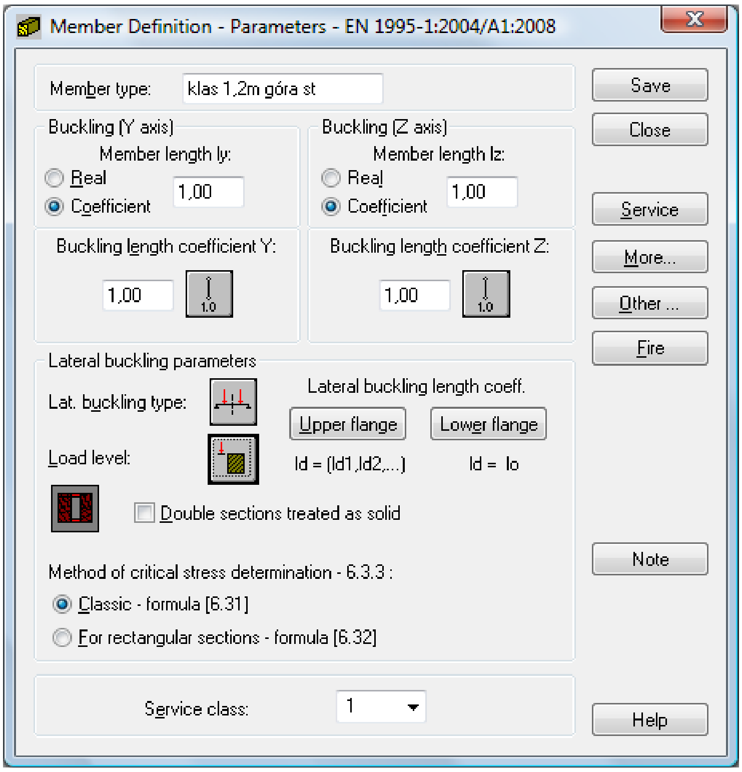


If you click the last icon - *Intermediate bracings* - the new dialog INTERNAL BRACINGS will appear.



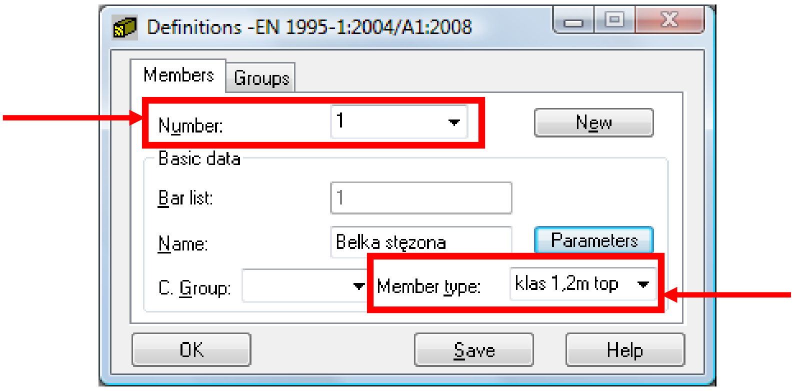
In the *INTERNAL BRACINGS* dialog, there are possibilities to define bracings for buckling and lateral buckling for the marked *member type* independently.
In this particular example of restrained elements, define member type with lateral buckling-upper flange internal bracings.

Save the newly-created member type under a new name:

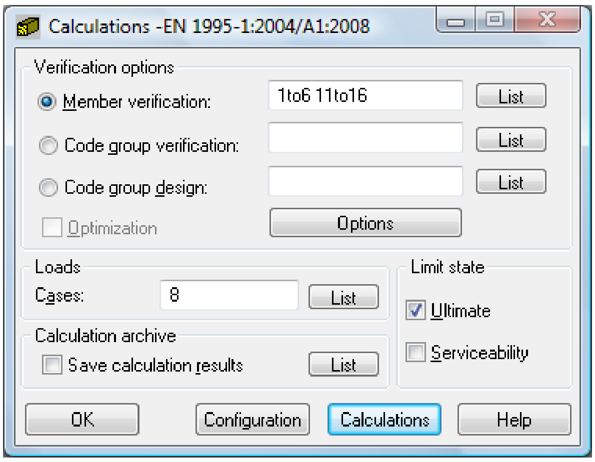


<u>Number</u> of the member must be assigned to the appropriate name of <u>*Member type*</u>.

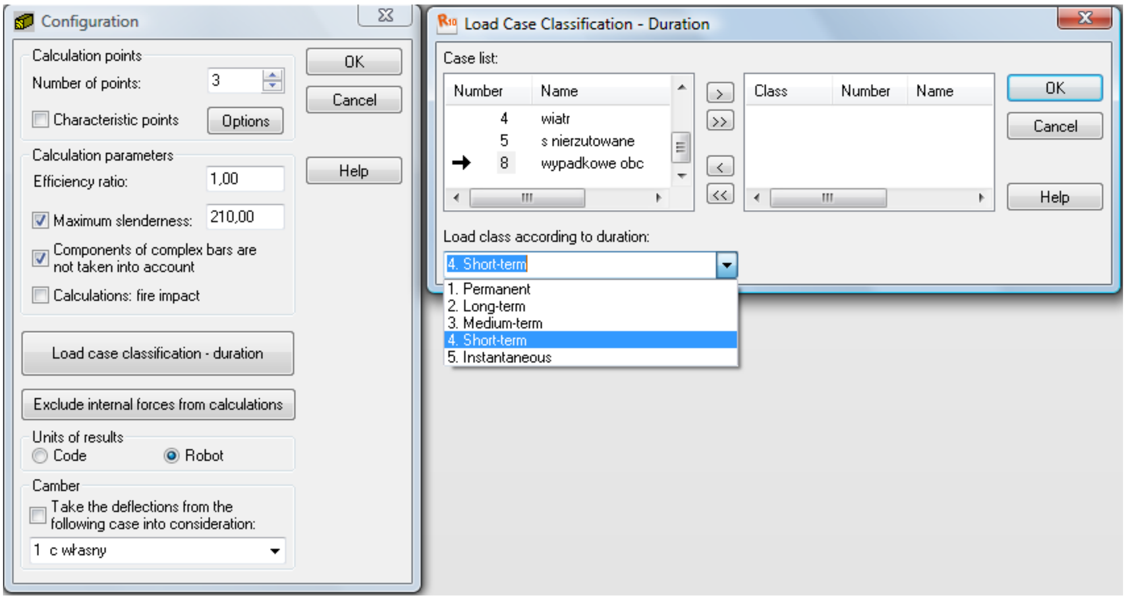
!!! It is very important when verifying different member types



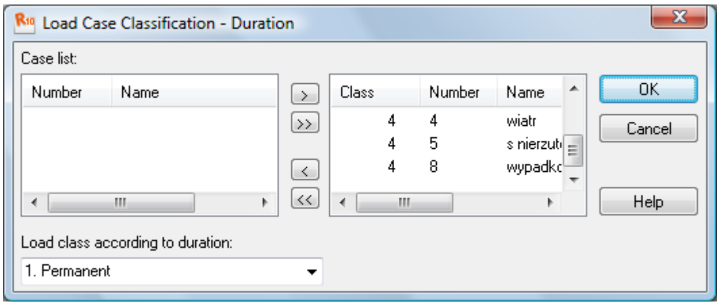
In the CALCULATIONS dialog set the following:
-> *Verification options - list of verified members*,
-> *Loads cases - list of chosen loads*
-> *Limit state*
->*Configuration*.



Before doing calculations you have to remember to specify appropriate duration for loads in the CALCULATIONS dialog:
- click [Configuration] button
- in CONFIGURATION dialog press [Load case classification - duration] button

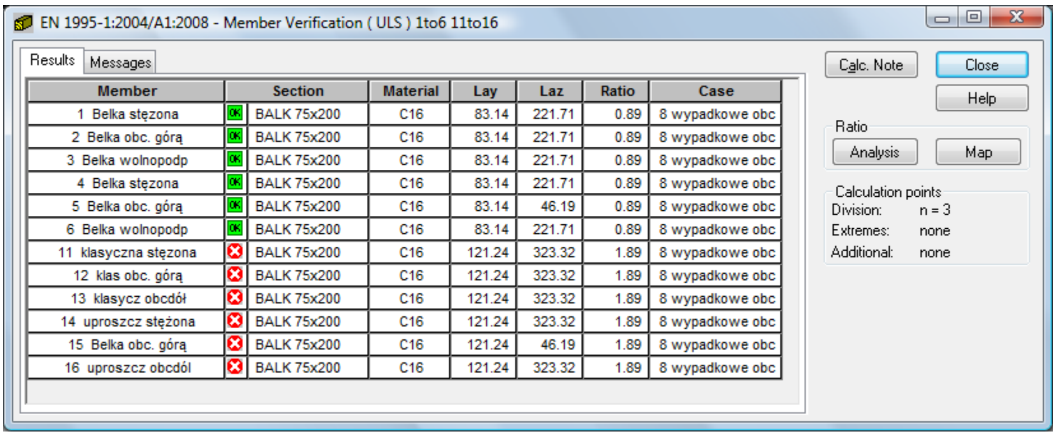


- in LOAD CASE CLASSIFICATION-DURATION dialog, assign “Load class according to duration” from selection list to the number of case list; for this particular example 4th “short-term” load case was selected and LOAD CASE CLASSIFICATION-DURATION dialog after the introduced changes looks as follows:



Follow up with the calculations now - press the *Calculations* button in the CALCULATIONS dialog.

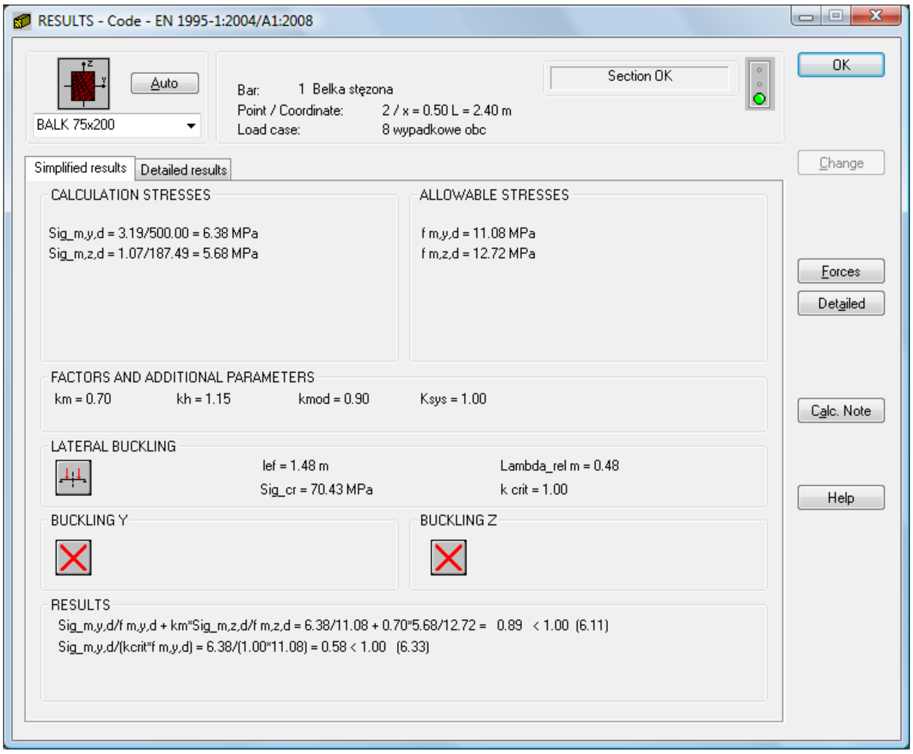
MEMBER VERIFICATION dialog with the most significant results data will appear on the screen.



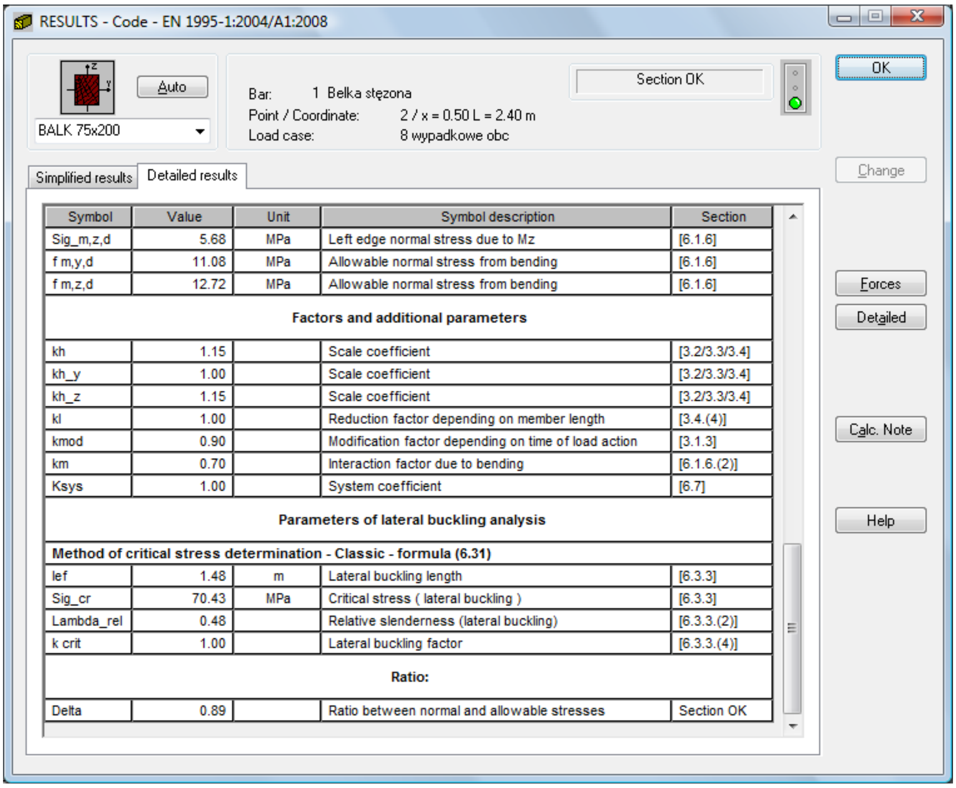
AUTODESK

Pressing the line with results for the member 1 opens the RESULTS dialog with detailed results for the analyzed member. The views of the RESULTS dialogs are presented below.

Simplified results tab



Detailed results tab



Symbol	Value	Unit	Symbol description	Section
Sig_m,z,d	5.68	MPa	Left edge normal stress due to Mz	[6.1.6]
f m,y,d	11.08	MPa	Allowable normal stress from bending	[6.1.6]
f m,z,d	12.72	MPa	Allowable normal stress from bending	[6.1.6]
			Factors and additional parameters	
kh	1.15		Scale coefficient	[3.2/3.3/3.4]
kh_y	1.00		Scale coefficient	[3.2/3.3/3.4]
kh_z	1.15		Scale coefficient	[3.2/3.3/3.4]
kl	1.00		Reduction factor depending on member length	[3.4.(4)]
kmod	0.90		Modification factor depending on time of load action	[3.1.3]
km	0.70		Interaction factor due to bending	[6.1.6.(2)]
Ksys	1.00		System coefficient	[6.7]
			Parameters of lateral buckling analysis	
Method of critical stress determination - Classic - formula (6.31)				
lef	1.48	m	Lateral buckling length	[6.3.3]
Sig_cr	70.43	MPa	Critical stress (lateral buckling)	[6.3.3]
Lambda_rel	0.48		Relative slenderness (lateral buckling)	[6.3.3.(2)]
k crit	1.00		Lateral buckling factor	[6.3.3.(4)]
			Ratio:	
Delta	0.89		Ratio between normal and allowable stresses	Section OK

Pressing the *Calc.Note* button in “RESULTS -Code” dialog opens the printout note for the analyzed member. You can obtain *Simplified results printout* or *Detailed results printout.* It depends on which tab is active. The printout note view of *Simplified results* is presented below.

RESULTS:

a) In the first step, BALK75x200 section was considered. The results are presented below.

TIMBER STRUCTURE CALCULATIONS

CODE: *EN 1995-1:2004/A1:2008*
ANALYSIS TYPE: Member Verification

CODE GROUP:
MEMBER: 1 Belka stężona **POINT:** 2 **COORDINATE:** x = 0.50 L = 2.40 m

---

LOADS:
Governing Load Case: 8 wypadkowe obc

MATERIAL C16

gM = 1.30	f m,0,k = 16.00 MPa	f t,0,k = 10.00 MPa	f c,0,k = 17.00 MPa
f v,k = 1.80 MPa	f t,90,k = 0.50 MPa	f c,90,k = 2.20 MPa	E 0,moyen = 8000.00 MPa
E 0,05 = 5400.00 MPa	G moyen = 500.00 MPa	Service class: 1	Beta c = 1.00

SECTION PARAMETERS: BALK 75x200

ht=20.0 cm

bf=7.5 cm	Ay=40.91 cm2	Az=109.09 cm2	Ax=150.00 cm2
tw=3.8 cm	Iy=5000.00 cm4	Iz=703.10 cm4	Ix=2148.0 cm4
tf=3.8 cm	Wely=500.00 cm3	Welz=187.49 cm3	

STRESSES	**ALLOWABLE STRESSES**
Sig_m,y,d = MY/Wy= 3.19/500.00 = 6.38 MPa	f m,y,d = 11.08 MPa
Sig_m,z,d = MZ/Wz= 1.07/187.49 = 5.68 MPa	f m,z,d = 12.72 MPa

---

Factors and additional parameters
km = 0.70 kh = 1.15 kmod = 0.90 Ksys = 1.00

LATERAL BUCKLING PARAMETERS:

lef = 1.48 m	Lambda_rel m = 0.48
Sig_cr = 70.43 MPa	k crit = 1.00

BUCKLING PARAMETERS:

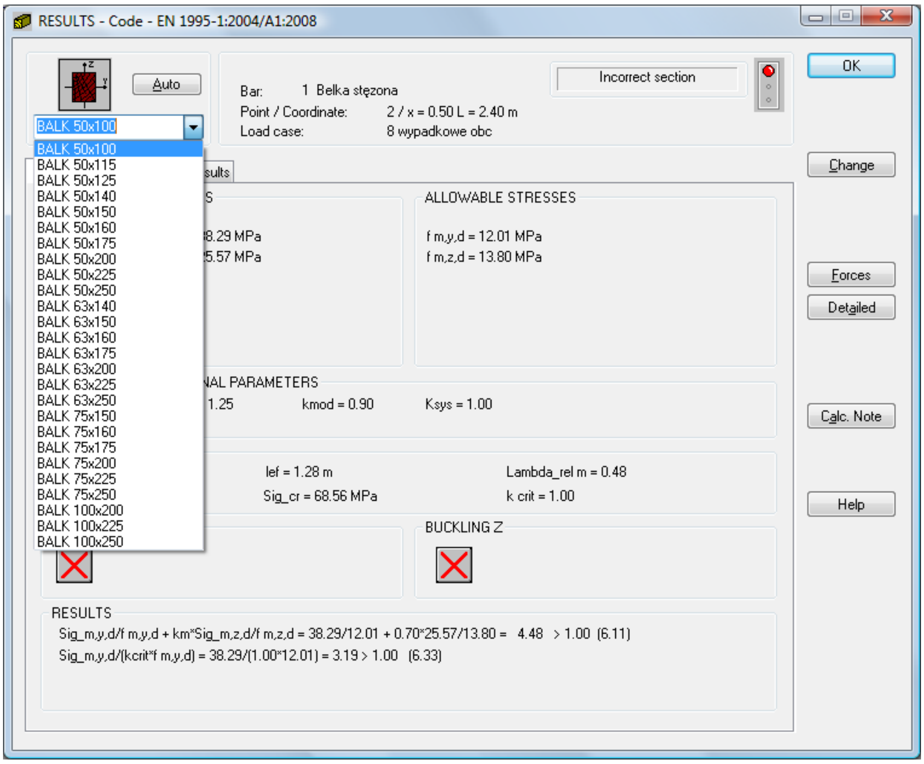
About Y axis: About Z axis:

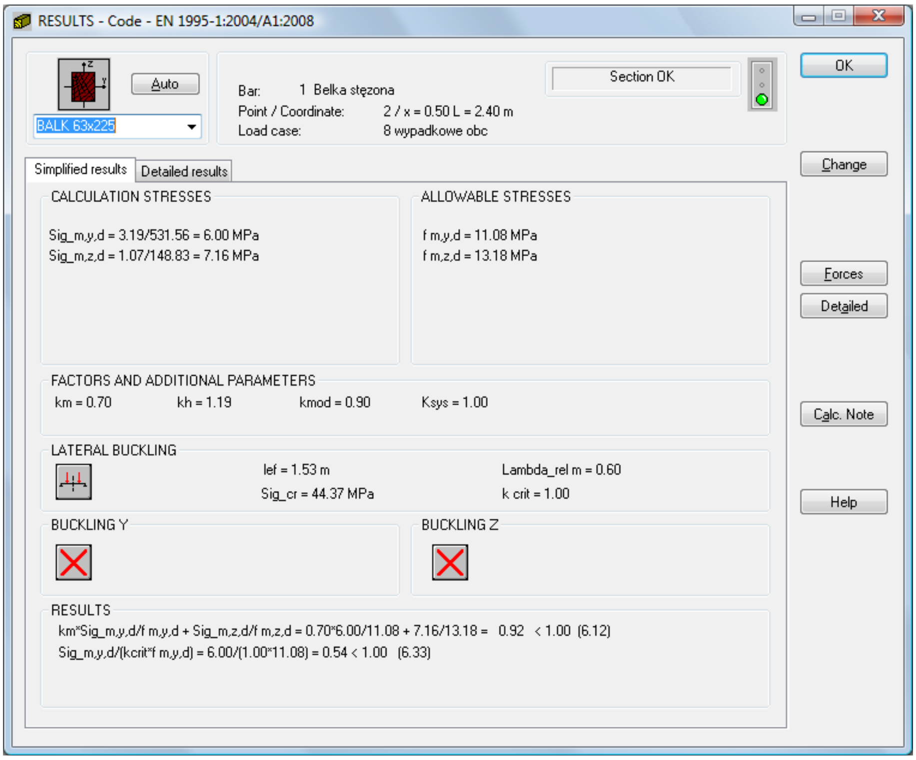
VERIFICATION FORMULAS:
Sig_m,y,d/f m,y,d + km*Sig_m,z,d/f m,z,d = 6.38/11.08 + 0.70*5.68/12.72 = 0.89 < 1.00 (6.11)
Sig_m,y,d/(kcrit*f m,y,d) = 6.38/(1.00*11.08) = 0.58 < 1.00 (6.33)

Section OK !!!

b) For economical reasons try to check the other, e.g. lighter BALK section.

While still in *RESULTS- CODE* dialog, type BALK only in the selection list and select the new section in the editable field, e.g. BALK 63x225. Press ENTER. Calculations and results are refreshed instantly.





The results for the newly selected section are presented below.

TIMBER STRUCTURE CALCULATIONS for BALK 63x225

CODE: *EN 1995-1:2004/A1:2008*
ANALYSIS TYPE: Member Verification

CODE GROUP:
MEMBER: 1 Belka stęzona **POINT:** 2 **COORDINATE:** x = 0.50 L = 2.40 m

---

LOADS:
Governing Load Case: 8 wypa dkowe obc

MATERIAL C16

gM = 1.30 f m,0,k = 16.00 MPa f t,0,k = 10.00 MPa f c,0,k = 17.00 MPa
f v,k = 1.80 MPa f t,90,k = 0.50 MPa f c,90,k = 2.20 MPa E 0,moyen = 8000.00 MPa
E 0,05 = 5400.00 MPa G moyen = 500.00 MPa Service class: 1 Beta c = 1.00

SECTION PARAMETERS: BALK 63x225

ht=22.5 cm
bf=6.3 cm Ay=31.02 cm2 Az=110.78 cm2 Ax=141.80 cm2
tw=3.1 cm Iy=5980.10 cm4 Iz=468.80 cm4 Ix=1544.5 cm4
tf=3.1 cm Wely=531.56 cm3 Welz=148.83 cm3

STRESSES
Sig_m,y,d = MY/Wy= 3.19/531.56 = 6.00 MPa
Sig_m,z,d = MZ/Wz= 1.07/148.83 = 7.16 Mpa

ALLOWABLE STRESSES
f m,y,d = 11.08 MPa
f m,z,d = 13.18 MPa

Factors and additional parameters
km = 0.70 kh = 1.19 kmod = 0.90 Ksys = 1.00

LATERAL BUCKLING PARAMETERS:
lef = 1.53 m Lambda_rel m = 0.60
Sig_cr = 44.37 MPa k crit = 1.00

BUCKLING PARAMETERS:
About Y axis: About Z axis:

VERIFICATION FORMULAS:
km*Sig_m,y,d/f m,y,d + Sig_m,z,d/f m,z,d = 0.70*6.00/11.08 + 7.16/13.18 = 0.92 < 1.00 (6.12)
Sig_m,y,d/(kcrit*f m,y,d) = 6.00/(1.00*11.08) = 0.54 < 1.00 (6.33)

Section OK !!!

COMPARISON for member n° 1 (BALK 75x200):

verification parameters, interaction expression		Robot	Handbook
L - beam length	[m]	4,8	4,8
Leff - effective length of the beam (Table 6.1, EC5)	[m]	1,48	1,48
σ m,cr = f (Leff) - critical bending stress	[MPa]	70,43	70,43
σ m,y,d - design bending stress due to My	[MPa]	6,382	6,39
σ m,z,d - design bending stress due to Mz	[MPa]	5,68	5,68
f m,y,d - design bending strength due to My	[MPa]	11,08	11,08
f m,z,d - design bending strength due to Mz	[MPa]	12,72	12,74
ratio (6.11) → σ m,y,d /f m,y,d + km* σ m,,z,d/f m,z,d =		0,889	0,89

CONCLUSIONS:

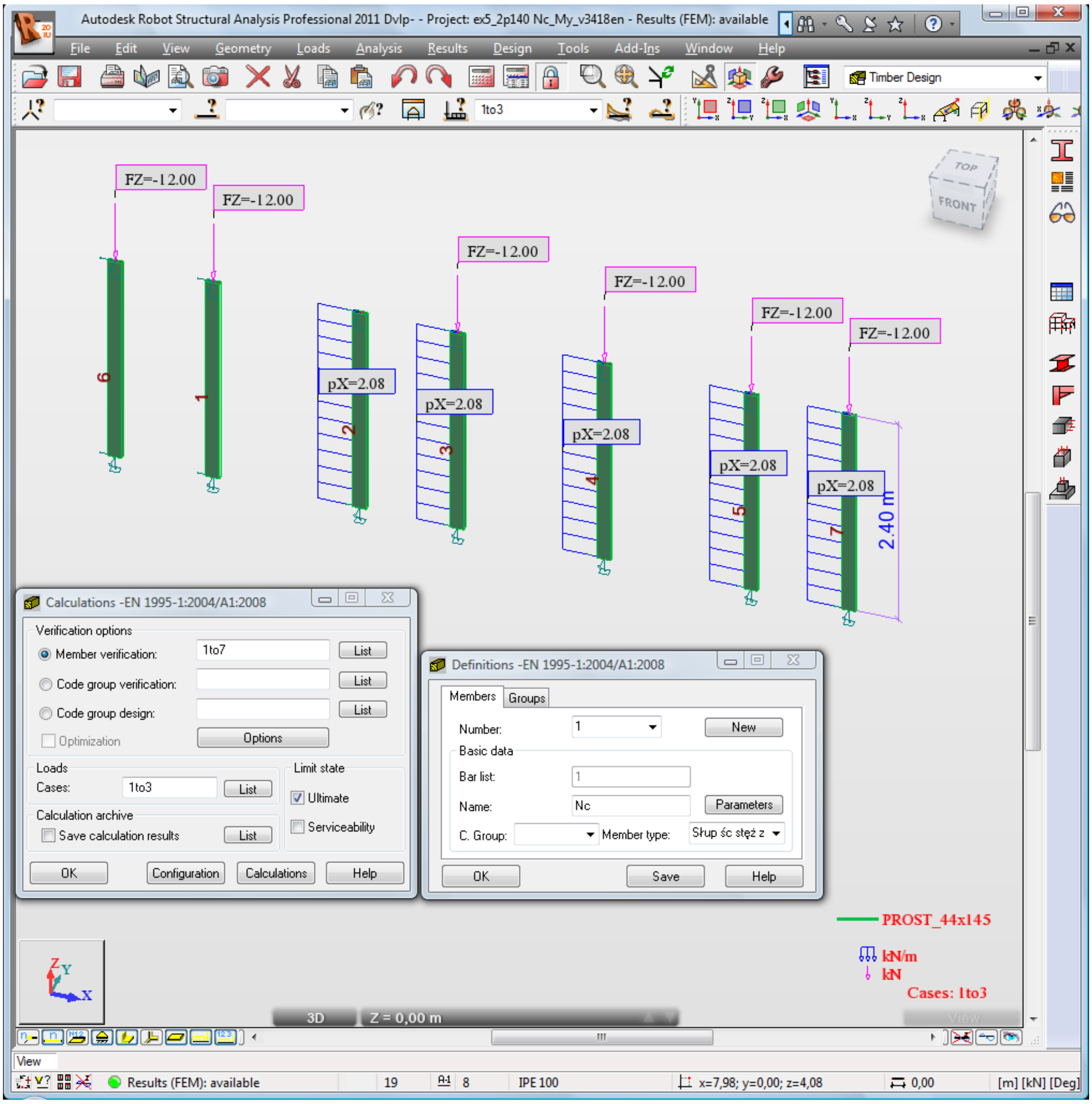
Agreement of results.
The small differences are caused by different accuracy of parameters in calculations.

VERIFICATION PROBLEM 2
combined compression and bending about one main axis

Example based on “Practical design of timber structures to Eurocode 5”
Hans Larsen and Vahik Enjily
File: EX_5_2p140_Nc_My.rtd

TITLE:
Example 5.2 - Solid Shape Subjected to Combined Compression and Bending About One Main Axis

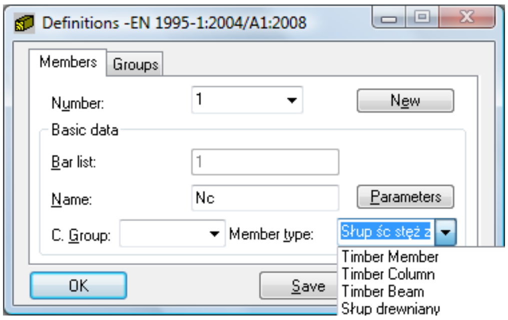
SPECIFICATION:
Verify if a simply supported rectangular columns of C16 with planed cross-section 44x145mm have sufficient available strength to support a permanent concentric compression load *Fz* = 12 kN and uniformly distributed lateral wind load inducing a design moment My = 1,5 kNm at mid-span about the strong axis. The unbraced length is 2,4m and Service Class 2. There are different types of buckling parameters for columns.



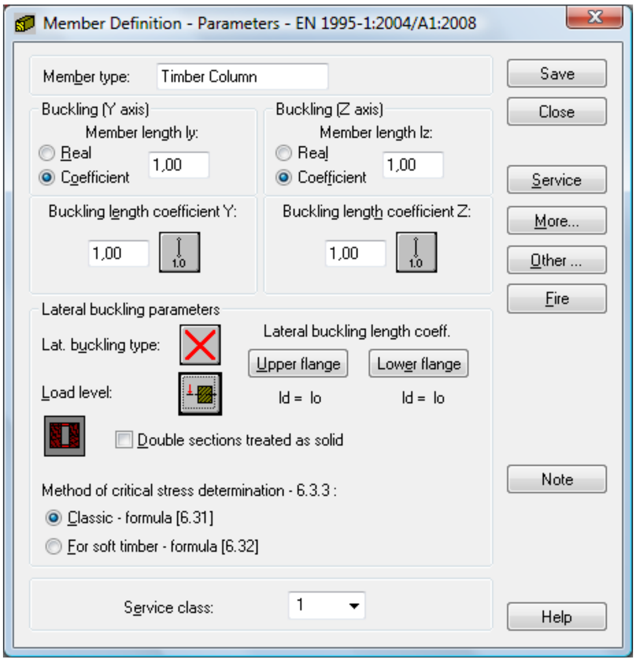
AUTODESK

SOLUTION:

After having defined and calculated the structure model, go to [Timber Design] tab.
In DEFINITIONS dialog, define a new type of member. It can be set in *Member type* combo-box.
Pre-defined type of member, e.g. "*timber column*" may be initially opened.

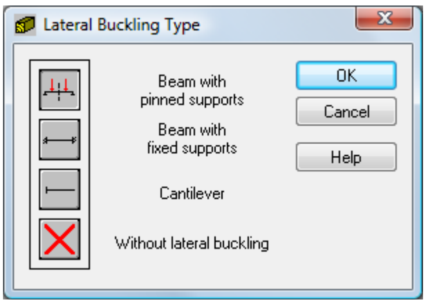


For the selected member type, press the *Parameters* button on *Members* tab.
The MEMBER DEFINITION-PARAMETERS dialog opens.

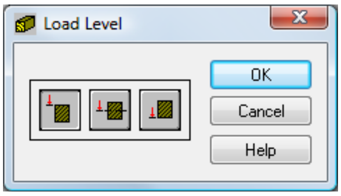


Type a new name in the *Member type* editable field. Next, change the parameters to meet the initial data requirements of a structure, e.g.:

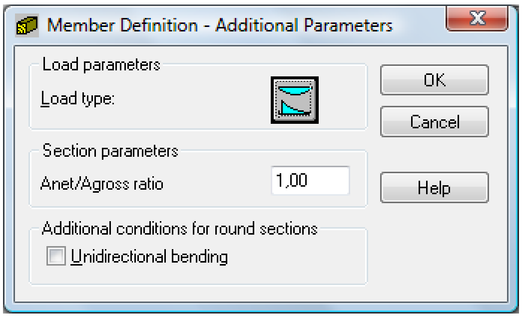
- switch on the appropriate *Lateral buckling type* icon*;*



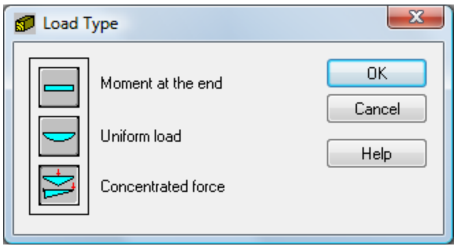
- select appropriate *Load level* icon



- define appropriate load type - press [More…] button; it opens ADDITIONAL PARAMETERS dialog



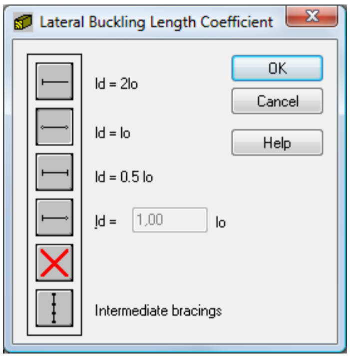
pressing the *Load type* icon opens a new dialog in which load type can be selected



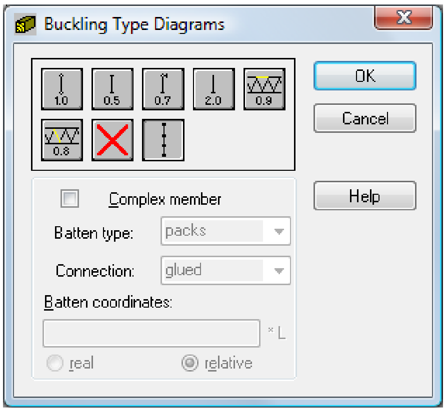
- define bracings for *Lateral buckling* and *Buckling.*

To define *Lateral buckling length coefficient* for a member, press *Upper/Lower flange* button or buckling type icon in [MEMBER DEFINITION-MEMBER] dialog.

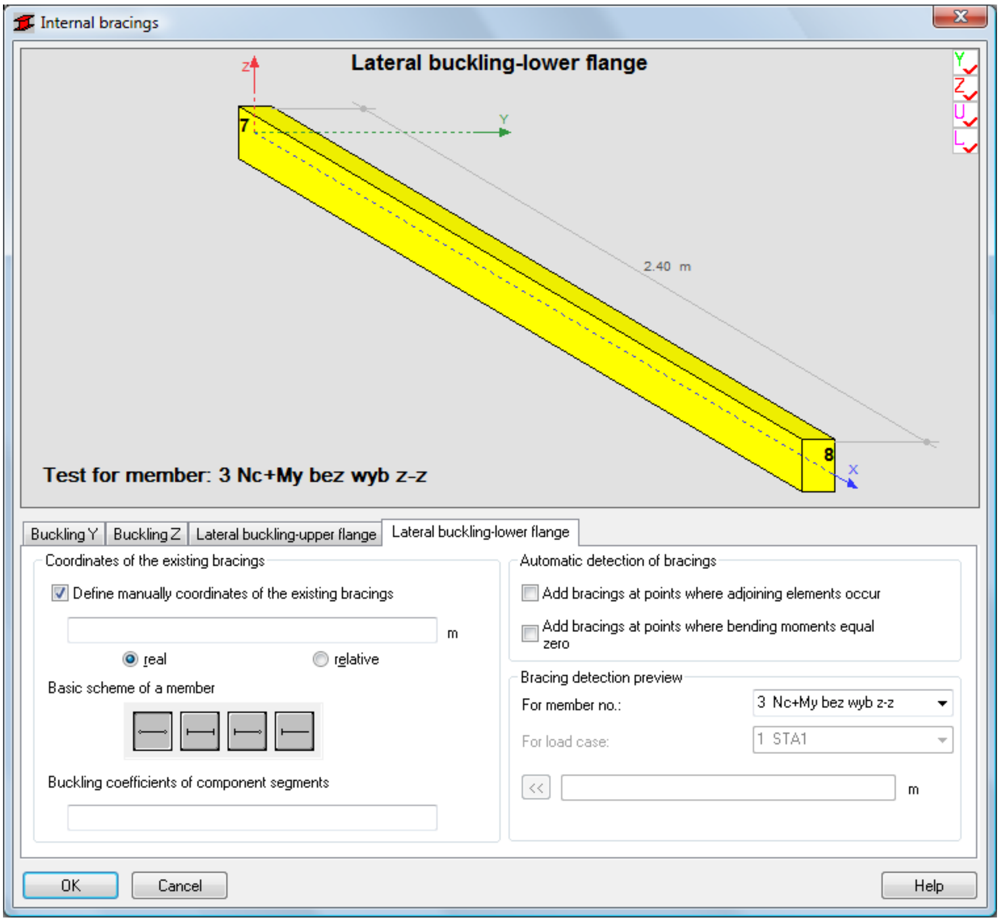
The first method opens LATERAL BUCKLING LENGTH COEFFICIENTS dialog,



the second opens > BUCKING TYPE DIAGRAMS dialog.

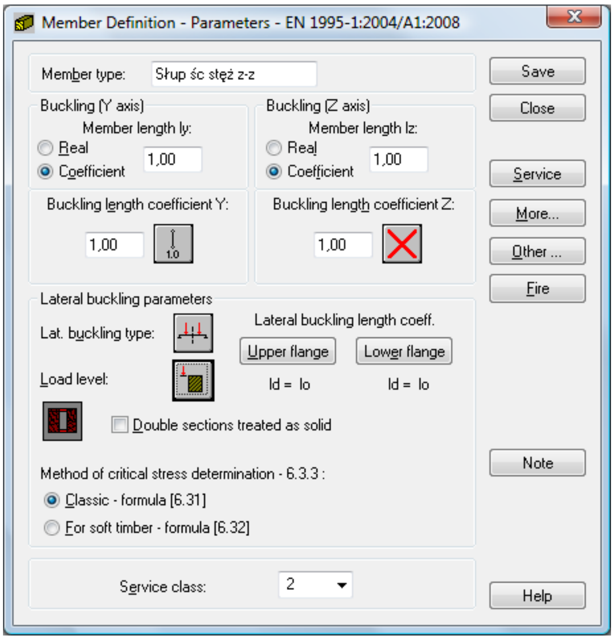


If you click the last icon *Intermediate bracings,* the new dialog INTERNAL BRACINGS will appear:



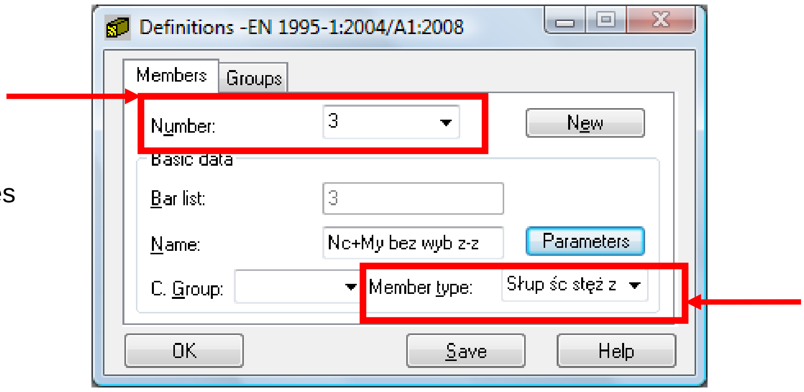
There are possibilities to define independently bracings for buckling and lateral buckling for the marked *member type* in *INTERNAL BRACINGS* dialog.

Save the newly-created member type under a new name.
The new MEMBER DEFINITION-PARAMETERS dialog defined for member n °3 verification looks as follows:

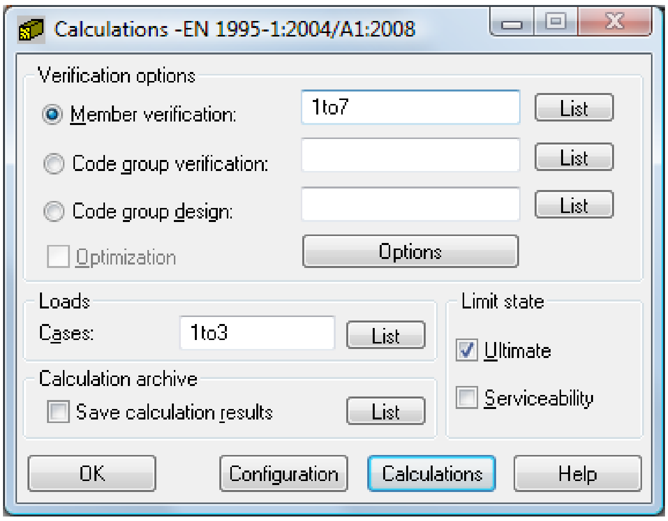


The *Number* of the member must be assigned to appropriate name of *Member type*

→ it is very important when verifying different member types



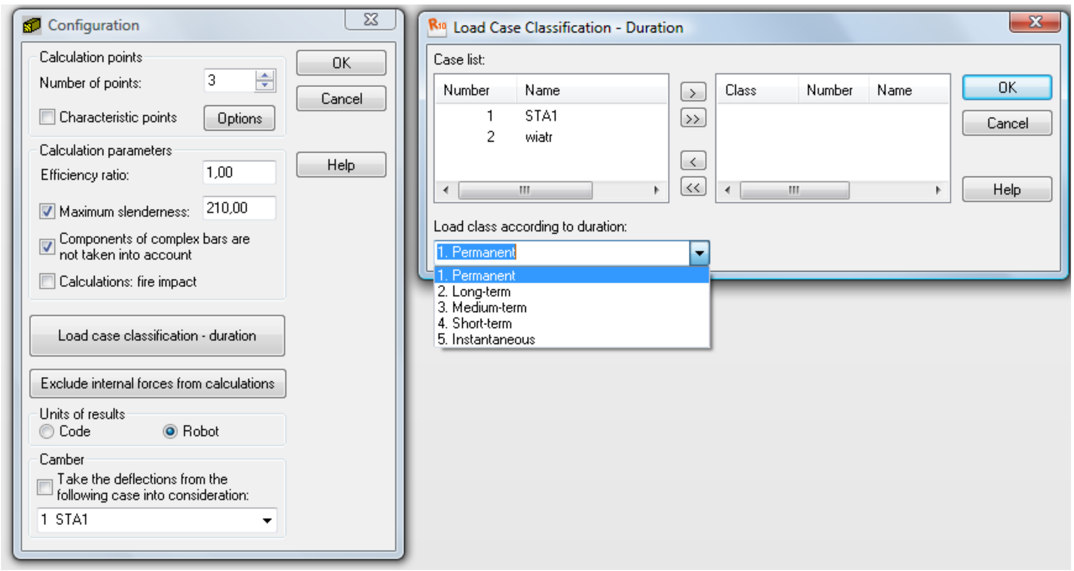
In CALCULATIONS dialog, set the following:
-> *Verification options - list of* verified *members*
-> *Loads cases - list of chosen loads*
-> *Limit state*
-> *Configuration*.



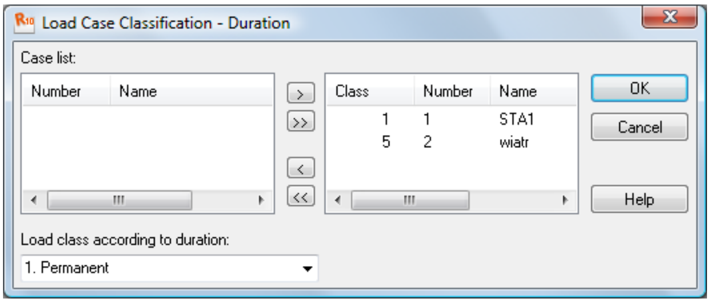
Before you verify the member, you have to specify appropriate duration for loads in CALCULATIONS dialog:

- click [Configuration] button

- in CONFIGURATION dialog, press [Load case classification - duration] button

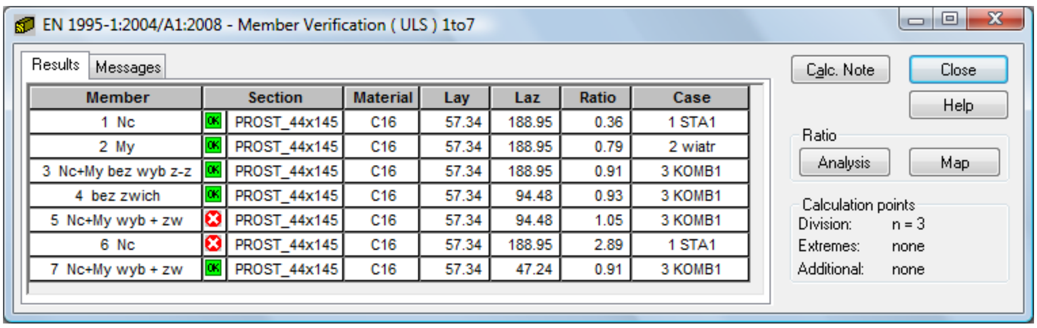


- in LOAD CASE CLASSIFICATION-DURATION dialog, assign "Load class according to duration" from combo box list to the number of the case list; in this particular example, the first "permanent" and the fifth "instantaneous" load case were selected and LOAD CASE CLASSIFICATION-DURATION dialog after the introduced changes looks as follows:



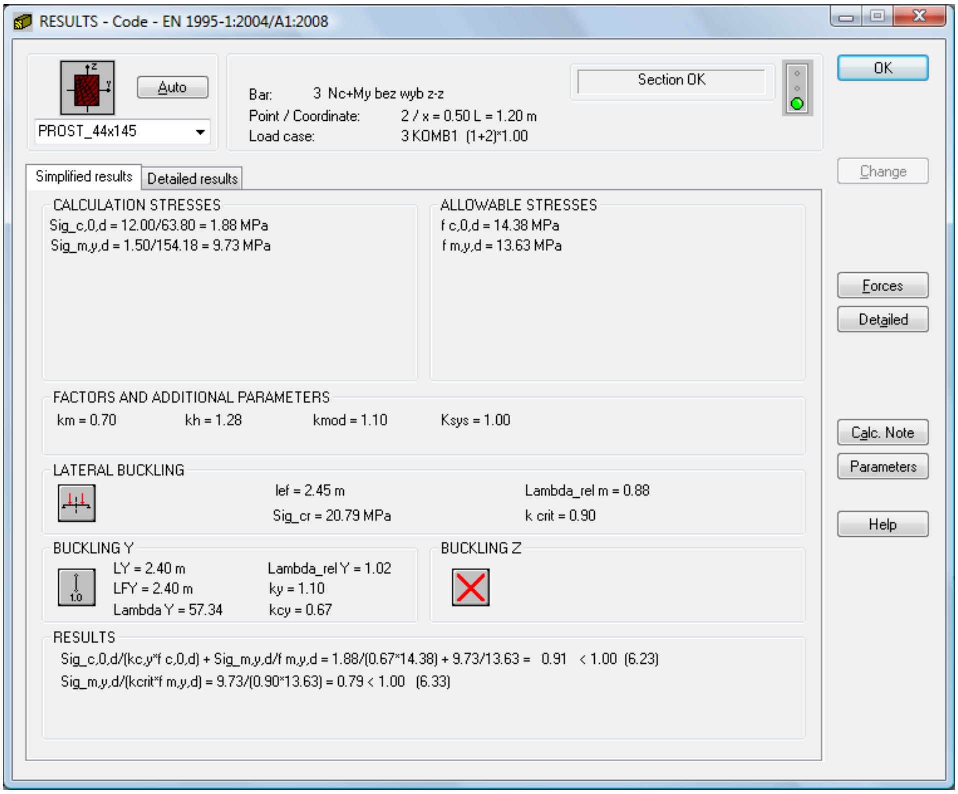
Start verification by pressing *Calculations* button in CALCULATIONS dialog.

MEMBER VERIFICATION dialog with most significant results data will appear on screen.

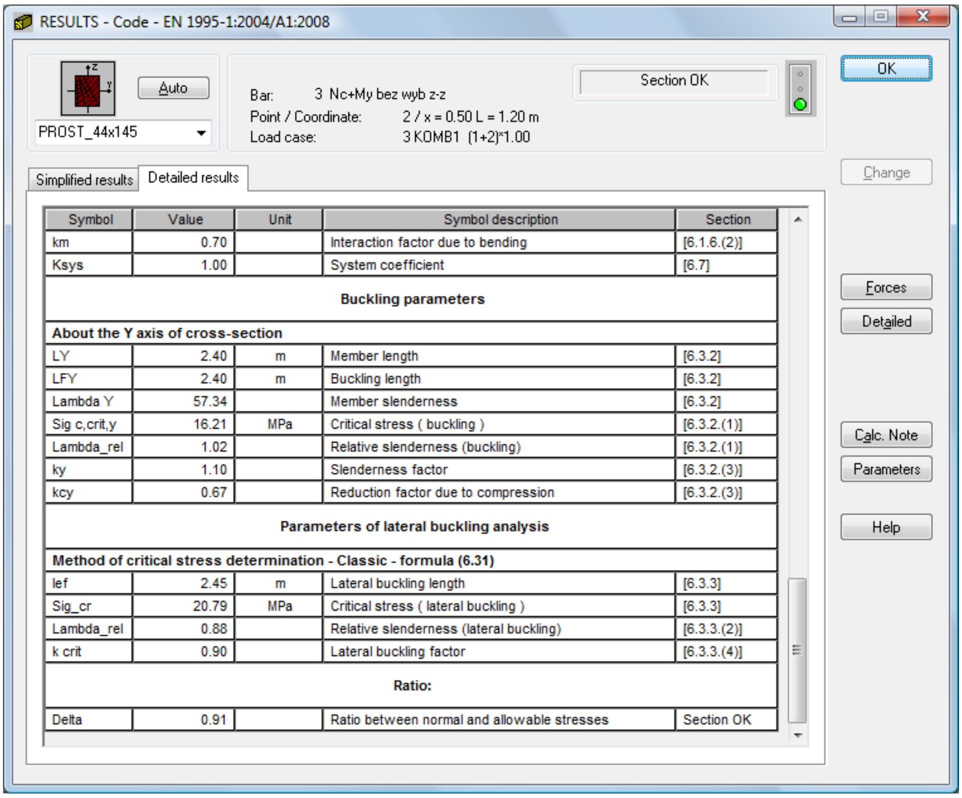


Pressing the line with the result for any member opens the RESULTS dialog with more detailed results for the analyzed member. The views of the RESULTS dialogs, e.g. for the third member, are presented below.

Simplified results tab



Detailed results tab



If you press the *Calc.Note* button in "RESULTS - Code" dialog, the printout note opens for the analyzed member. You can obtain *Simplified results printout* or *Detailed results printout*. It depends on which tab is active. The printout note view of *Simplified results* is presented below.

RESULTS:

TIMBER STRUCTURE CALCULATIONS

CODE: *EN 1995-1:2004/A1:2008*
ANALYSIS TYPE: Member Verification

CODE GROUP:
MEMBER: 3 Nc+My bez wyb z-z **POINT:** 2 **COORDINATE:** x = 0.50 L = 1.20 m

---

LOADS:
Governing Load Case: 3 KOMB1 (1+2)*1.00

MATERIAL C16

gM = 1.30	f m,0,k = 16.00 MPa	f t,0,k = 10.00 MPa	f c,0,k = 17.00 MPa
f v,k = 1.80 MPa	f t,90,k = 0.50 MPa	f c,90,k = 2.20 MPa	E 0,moyen = 8000.00 MPa
E 0,05 = 5400.00 MPa	G moyen = 500.00 MPa	Service class: 2	Beta c = 0.20

SECTION PARAMETERS: PROST_44x145

ht=14.5 cm			
bf=4.4 cm	Ay=14.85 cm2	Az=48.95 cm2	Ax=63.80 cm2
tw=2.2 cm	Iy=1117.83 cm4	Iz=102.93 cm4	Ix=333.0 cm4
tf=2.2 cm	Wely=154.18 cm3	Welz=46.79 cm3	

STRESSES	**ALLOWABLE STRESSES**
Sig_c,0,d = N/Ax = 12.00/63.80 = 1.88 MPa	f c,0,d = 14.38 MPa
Sig_m,y,d = MY/Wy= 1.50/154.18 = 9.73 MPa	f m,y,d = 13.63 MPa

---

Factors and additional parameters
km = 0.70 kh = 1.28 kmod = 1.10 Ksys = 1.00

LATERAL BUCKLING PARAMETERS:

lef = 2.45 m	Lambda_rel m = 0.88
Sig_cr = 20.79 MPa	k crit = 0.90

BUCKLING PARAMETERS:

About Y axis: About Z axis:

LY = 2.40 m	Lambda Y = 57.34
Lambda_rel Y = 1.02	ky = 1.10
LFY = 2.40 m	kcy = 0.67

VERIFICATION FORMULAS:
Sig_c,0,d/(kc,y*f c,0,d) + Sig_m,y,d/f m,y,d = 1.88/(0.67*14.38) + 9.73/13.63 = 0.91 < 1.00 (6.23)
Sig_m,y,d/(kcrit*f m,y,d) = 9.73/(0.90*13.63) = 0.79 < 1.00 (6.33)

Section OK !!!

COMPARISON:

e.g. for member n º 3 → for the axial load Nc and My moment

verifications parameters, interaction expression	Robot	Handbook
λy - member slenderness	57,34	57,3
ky - slenderness factor	1,097	1,097
kcy - reduction factor due to compression	0,671	0,671
k_{mod}	1,1	1,1
f c,o,d - design compression strength [MPa]	14,38	14,38
f m,y,d - design bending strength due to My [MPa]	13,63	13,54
σ c,o,d - design compression stress [MPa]	1,88	1,88
σ m,y,d - design bending stress due to My [MPa]	9,73	9,73
ratio from (6.23) → σ c,o,d / ($k_{c,y}$ *fc,o,d) + σ m,y,d / f m,y,d =	**<u>0,91</u>**	**<u>0,91</u>**

CONCLUSIONS:

Total agreement of results.