Robot Structural Analysis Professional

AUTODESK

VERIFICATION MANUAL FOR EUROCODES

www.autodesk.com/robot-structural-analysis

Copyright © 2024 Autodesk, Inc.

INTRODUCTION	3
STEEL	4
1. EUROCODE 3 (EN 1993-1-1:2005)	5
VERIFICATION EXAMPLE 1 - AXIAL COMPRESSION VERIFICATION EXAMPLE 2 - AXIAL COMPRESSION WITH BUCKLING	9
VERIFICATION EXAMPLE 3 - COMBINED COMPRESSION AND BENDING VERIFICATION EXAMPLE 4 - BENDING WITH LATERAL BUCKLING VERIFICATION EXAMPLE 5 - COMBINED BI-AXIAL BENDING AND COMPRESSION	15
CONCRETE	23
1. EUROCODE 2 EN 1992-1-1:2004 AC:2008 - RC BEAMS	24
VERIFICATION EXAMPLE 1 - DIMENSIONING REINFORCEMENT IN RECTANGULAR SECTION AT BENDING VERIFICATION EXAMPLE 2 - DIMENSIONING REINFORCEMENT IN RECTANGULAR SECTION AT BENDING VERIFICATION EXAMPLE 3 - DIMENSIONING REINFORCEMENT IN RECTANGULAR SECTION AT BENDING WITH COMPRESSION VERIFICATION EXAMPLE 4 - DIMENSIONING REINFORCEMENT IN RECTANGULAR SECTION AT BENDING WITH COMPRESSION VERIFICATION EXAMPLE 5 - DIMENSIONING OF SHEAR REINFORCEMENT IN BEAM WITH RECTANGULAR SECTION VERIFICATION EXAMPLE 5 - DIMENSIONING OF SHEAR REINFORCEMENT IN BEAM WITH RECTANGULAR SECTION VERIFICATION EXAMPLE 6 - DEFLECTION OF SIMPLY SUPPORTED BEAM WITH RECTANGULAR SECTION LITERATURE	29 31 33 35 40
2. EUROCODE 2 EN 1992-1-1:2004 AC:2008 - RC COLUMNS	43
VERIFICATION EXAMPLE 1 - COLUMN SUBJECTED TO AXIAL LOAD AND UNI-AXIAL BENDING	
3. EUROCODE 2 EN 1992-1-1:2004 AC:2008 - RC SLABS (PUNCHING)	51
VERIFICATION PROBLEM 1 - PUNCHING CAPACITY OF SLAB WITHOUT SHEAR REINFORCEMENT VERIFICATION PROBLEM 2 - PUNCHING CAPACITY OF SLAB WITHOUT SHEAR REINFORCEMENT FOR FINNISH NAD VERIFICATION PROBLEM 3 - CALCULATION OF PUNCHING FORCE FOR ECCENTRICALY APPLIED SUPPORT REACTION VERIFICATION PROBLEM 4 - PUNCHING CAPACITY OF SLAB WITH SHEAR REINFORCEMENT LITERATURE	56 58 61
TIMBER	68
1. EUROCODE 5: DESIGN OF TIMBER STRUCTURES; EN 1995-1:2004/A1:2008	69
GENERAL REMARKS	72



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



Autodesk [and other products] are registered trademarks or trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and/or other countries. All other brand names, product names, or trademarks belong to their respective holders. Autodesk reserves the right to alter product and services offerings, and specifications and pricing at any time without notice, and is not responsible for typographical or graphical errors that may appear in this document. © 2024 Autodesk, Inc. All rights reserved.

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



VERIFICATION EXAMPLE 1 - Axial compression

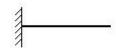
Example taken from Designer's Guide to EN 1993-1-1 L.Gardner and D.A.Nethercot, Thomas Telford Publishing, 2005

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.

🗾 Definitions - EN 1993-1:2005	🖆 Member Definition - Parameters - EN 1993-1:2005 🛛 🗙
Members Groups	Membertype: Column1 Save
Number: 1 New Basic data	Buckling (Y axis) Close Member length ly: Member length lz: Beal 1.00 Coefficient 1.00 Buckling length coeff. Y: Buckling length coeff. Z: 1.00 Non-sway Buckling curve Y auto Elexural-torsional buckling of monosymmetric sections
Image: space with the point of the poi	Lateral buckling parameters More Lateral buckling Lateral buckling length coefficient Load level: ⊥ Lor = lo Lcr = lo C General method [6.3.2.2] Lambda LT,0 = Other 0.75 Simplified method for beams with kfl = Iateral restraints [5.3.2.4] Additional sets of member parameters ✓ Limit deflections and displacements: Service Complex sections:
	Image: Inin-walled sections: Thin-walled
	Eire Help

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.



Autodesk [and other products] are registered trademarks or trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and/or other countries. All other brand names, product names, or trademarks belong to their respective holders. Autodesk reserves the right to alter product and services offerings, and specifications and pricing at any time without notice, and is not responsible for typographical or graphical errors that may appear in this document. © 2024 Autodesk, Inc. All rights reserved.

🗲 Calculations - EN 1993-1	:2005
Verification options	
• Member verification:	1 List
C Code group verification:	List
C Code group <u>d</u> esign:	List
<u>Optimization</u>	Options
Loads	Limit state
C <u>a</u> ses: 1	List 🔽 Ultimate
Calculation archive	List Serviceability
OK Configu	ration Calculations Help

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.

1	EN 1993-1:2005 - Member Verification (ULS) 1									
	Results Messages									<u>C</u> lose
	Member	Section	Material	Lay	Laz	Ratio	Case			Help
	1 Bar 1	UC 254x254x7	S355	36.13	61.74	1.00	1 STA1		r Ratio	
									Analysis	Map
									Calculation po Division:	n = 3
										none
									Additional:	none

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.

💯 RESULTS - Code - EN 1993-1:2005		🗲 RESULTS - Co	de - EN 1993-1:	2005			
Auto Bar: 1 Bar 1 UC 254x254x73 V Section 0K If a = 0.00 L = 0.00 m	<u> 0K </u>	UC 254x254x73	<u>Auto</u>	Bar: Point / Coo Load case:			<u> </u>
Simplified results Detailed results	Change	Simplified results	Detailed results	1			Dhange
FORCES N.Ed = 3305 kN		Symbol	Values	Unit	Symbol description Section		
Nc,Rd = 3305 kN Nb Rd = 3305 kN				Cross-	section properties: UC 254x254x73		
	Eorces	Ax	93.100	cm2	Cross-section area	-	Eorces
	Dublid	Ay	75.883	cm2	Shear area - Y-axis		Detailed
Class of section = 2	Detgled	Az	25.792	cm2	Shear area - Z-axis		Degieo
LATERAL BUCKLING		lx .	57.600	cm4	Torsional constant	7 – 1	
×LT = 1.00		ly	11410.000	cm4	Moment of inertia of a section about the Y-axis		
		Iz	3908.000	cm4	Moment of inertia of a section about the Z-axis		
BUCKLING Z	Calc. Note	Wply	990.000	cm3	Plastic section modulus about the Y (major) axis		Calc. Note
	- cgic. Hoto	Wplz	463.000	cm3	Plastic section modulus about the Z (minor) axis		- Cgic. Hoto
	Parameters	h	25.40	cm	Height of cross-section		Parameters
		b	25.40	cm	Width of cross-section		
	Help	tf	1.42	cm	Flange thickness		Help
- SECTION CHECK		tw	0.86	cm	Web thickness		
N.Ed/Nc.Rd = 1.00 < 1.00 (6.2.4.(1))		ry	11.07	cm	Radius of gyration - Y-axis		
		rz	6.48	cm	Radius of gyration - Z-axis		
MEMBER STABILITY CHECK					Material:		
Not analyzed		Name			\$355 (\$355)		
		fy	355.00	MPa	Design yield strength of material (3.2)	-	
						_	

STEEL DESIGN

CODE GROUP: MEMBER: 1 Bar 1 0.00 m	POINT: 1	cc	DORDINATE: $x = 0.00 L =$
LOADS: Governing Load Case	: 1 STA1		
MATERIAL: S355 (S355) fy =	= 355.00 MPa		
	PARAMETERS: UC 254x254	4x73	
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 tf=1.42 cm	Iy=11410.000 cm4 Wply=990.000 cm3	Iz=3908.000 cm4	Ix=57.600 cm4
u=1.42 cm	wpry=990.000 cm3	Wplz=463.000 cm3	
N,Ed = 3305 kN Nc,Rd = 3305 kN	S AND CAPACITIES:		
Nb,Rd = 3305 kN			Class of section $= 2$
	UCKLING PARAMETERS:		
	METERS:		
About Y axis	5	About Z axis:	

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

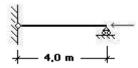
Example taken from Designer's Guide to EN 1993-1-1 L.Gardner and D.A.Nethercot, Thomas Telford Publishing, 2005

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.

🗾 Definitions - EN 1993-1:2005	🗲 Member Definition - Parameters - EN 1993-1:2005 🛛 🗙
Members Groups	Member type: Column2 Save
Number: 1 New Basic data Bar list: 1 Name: Bar 1 Parameters C. Group: Member type: Column1 OK Save Help	Buckling (Y axis) Member length ly: Member length lz: Me
Image: space with the systemImage: space with the system	Lateral buckling parameters Lateral buckling length coefficient Load level: Upper flange Lower flange Load level: Lor = lo Lor = lo C General method [6.3.2.2] Lambda LT,0 = 0.40 © Detailed method [6.3.2.3] Beta = 0.75 Simplified method for beams with kfl = 1.10
• Non-sway structure	Limit deflections and displacements: Service Complex sections:
	Thin-walled sections: Thin-walled Note
	Eire Help

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.



Autodesk [and other products] are registered trademarks or trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and/or other countries. All other brand names, product names, or trademarks belong to their respective holders. Autodesk reserves the right to alter product and services offerings, and specifications and pricing at any time without notice, and is not responsible for typographical or graphical errors that may appear in this document. © 2024 Autodesk, Inc. All rights reserved.

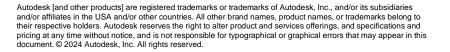
🗲 Calculations - EN 1993-1	:2005
Verification options	
• Member verification:	1 List
C Code group verification:	List
C Code group <u>d</u> esign:	List
Dptimization	Options
Loads	Limit state
C <u>a</u> ses: 1	List 🔽 Ultimate
Calculation archive	
Save calculation results	
OK Configu	Iration Calculations Help

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.

1	🗲 EN 1993-1:2005 - Member Verification (ULS) 1									
	Results Messages									<u>Close</u>
I	Member	Section	Material	Lay	Laz	Ratio	Case			Help
	1 Bar 1	CHS 244.5x10	S275	48.21	48.21	0.89	1 STA1		r Ratio	
									Analysis	Map
									Calculation po Division:	n = 3
										none
									Additional:	none

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.

💯 RESULTS - Code - EN 1993-1:2005	S RESULTS	- Code - EN 1993-1	:2005		,		
▲ulo Bar 1 Bar 1 Section 0K Section 0K CHS 244.5x10 Image: Section 0K 1 / x = 0.00 L = 0.00 m Load case: 1 STA1	<u> </u>	CHS 244.5		Bar: Point / Co Load case			<u> </u>
Simplified results Detailed results	Change	Simplified r	esults Detailed result	5			Change
FORCES N.E.d = 1630.0 kN		Symb	ol Values	Unit	Symbol description Secti	on 🔺	1
Nc.Rd = 2026.8 kN Nb.Rd = 1836.5 kN				Cross	s-section properties: CHS 244.5x10		
	Eorces	Ax	73.700	cm2	Cross-section area		Eorces
		Ay	46.919	cm2	Shear area - Y-axis		
Class of section = 1		Az	46.919	cm2	Shear area - Z-axis		
LATERAL BUCKLING		bx .	10150.000	cm4	Torsional constant		
XLT = 1.00		ly	5073.000	cm4	Moment of inertia of a section about the Y-axis		
		IZ	5073.000	cm4	Moment of inertia of a section about the Z-axis		
F BUCKLING Y	Calc. Note	Wply	550.236	cm3	Plastic section modulus about the Y (major) axis		Calc. Note
Ly=4.00 m Lam_y=0.56 Lz=4.00 m Lam_z=0.56		Wplz	550.236	cm3	Plastic section modulus about the Z (minor) axis		
10 Lor,y = 4.00 m Xy = 0.91 10 Lor,z = 4.00 m Xz = 0.91	Parameters	h	24.45	cm	Height of cross-section		Parameters
Lamy = 48.21		b	24.45	cm	Width of cross-section		
	Help	tf	1.00	cm	Flange thickness		Help
SECTION CHECK	nep	tw	1.00	cm	Web thickness		nep
NEd/Nc,Rd = 0.80 < 1.00 (6.2.4.(1))		ry	8.30	cm	Radius of gyration - Y-axis		
		rz	8.30	cm	Radius of gyration - Z-axis		
MEMBER STABILITY CHECK Lamv = 48 21 < Lam.max = 210.00 Lamz = 48.21 < Lam.max = 210.00 STABLE					Material:		
N,Ed/Nb,Rd = 0.89 < 1.00 (6.3.1.1.(1))		Name			\$275 (\$275)		
		fy	275.00	MPa	Design yield strength of material (3.2)	-	
							_



AUTODESK

STEEL DESIGN

CODE: EN 1993-1:2 ANALYSIS TYPE: M	2005, Eurocode 3: Designment of the second state of the second sta	gn of steel structures.	
CODE GROUP: MEMBER: 1 Bar 1 0.00 m	POINT: 1	со	ORDINATE: x = 0.00 L =
LOADS: Governing Load Case:	1 STA1		
MATERIAL: S275 (S275) fy = 2	75.00 MPa		
SECTION PA h=24.45 cm b=24.45 cm tw=1.00 cm tf=1.00 cm	RAMETERS: CHS 244. gM0=1.00 Ay=46.919 cm2 Iy=5073.000 cm4 Wply=550.236 cm3	5x10 gM1=1.00 Az=46.919 cm2 Iz=5073.000 cm4 Wplz=550.236 cm3	Ax=73.700 cm2 Ix=10150.000 cm4
INTERNAL FORCES N,Ed = 1630.0 kN Nc,Rd = 2026.8 kN Nb,Rd = 1836.5 kN	AND CAPACITIES:		Class of section = 1
	CKLING PARAMETER	S:	
BUCKLING PARAME \downarrow_{10} About Y a Ly = 4.00 m Lcr, y = 4.00 m Lamy = 48.21	-	Lz = 4.00 m Lcr,z = 4.00 m Lamz = 48.21	tis: Lam_z = 0.56 Xz = 0.91
VERIFICATION FORM Section strength check: N,Ed/Nc,Rd = 0.80 < 1.0 Global stability check of Lambda,y = 48.21 < Lan N,Ed/Nb,Rd = 0.89 < 1.0	00 (6.2.4.(1)) f <i>member:</i> nbda,max = 210.00 I	Lambda,z = 48.21 < Lambda,m	ax = 210.00 STABLE
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.

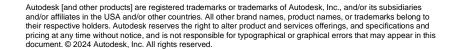


SOLUTION:

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.

🗾 Definitions - EN 1993-1:2005	💋 Member Definition - Parameters - EN 1993-1:2005	2
Members Groups	Member type: Column1	Save
Number: 1 New Basic data	Buckling (Y axis) Buckling (Z axis) Member length ly: Member length ly: Beal 1.00 Coefficient 1.00 Buckling length coeff. Y: Doefficient Non-sway Buckling curve Y Buckling curve Y auto Elexural-torsional buckling of monosymmetric sections	Close
Image: Constraint of the second state of the second st	Lateral buckling parameters Lateral buckling length coefficient Load level: Upper flange Lor = lo Lor = lo C general method [6.3.2.2] Lambda LT,0 = 0.40 C general method [6.3.2.3] Beta = 0.75 Simplified method for beams with lateral restraints [5.3.2.4] kfl = Additional sets of member parameters Service C Lomplex sections: Complex Thin-walled sections: Thin-walled	More
		Note Help

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.





🗲 Calculations - EN 1993-1	:2005
Verification options	
• Member verification:	1 List
C Code group verification:	List
C Code group <u>d</u> esign:	List
Dptimization	Options
Loads	Limit state
C <u>a</u> ses: 1	List 🔽 Ultimate
Calculation archive	
Save calculation results	
OK Configu	Iration Calculations Help

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.

1	EN 1993-1:2005 - Member Verification (ULS) 1									
	Results Messages	C <u>a</u> lc. Note	<u>Close</u>							
	Member	Section	Material	Lay	Laz	Ratio	Case			Help
	1 Bar 1	UB 457x191x9	S235	20.91	92.31	1.00	1 STA1		r Ratio	
									Analysis	Map
									Calculation po Division:	n = 3
										none
									Additional:	none

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.

FRESULTS - Code - EN 1993-1:2005	_ 🗆 🗵	FRESULTS	- Code - EN 1993-1	:2005			_ 🗆 🗡
UB 457x191x38 X UB 457x191x38 X UB 457x191x38 X	<u> </u>	UB 457x1		Bar: Point / Cor Load case			<u> ОК</u>
Simplified results Detailed results	Change	Simplified r	esults Detailed result	s]			Dhange
FORCES N.Ed = 1400.0 kN My.Ed = -342.2 kN*m	1	Symb		Unit	Symbol description Section		
Nc,Rd = 2937.5 kN My,pLRd = 525.0 kN*m Nb,Rd = 2937.5 kN My,c,Rd = 525.0 kN*m				Cross	-section properties: UB 457x191x98		
My.N.Rd = 342.5 kN*m	Eorces	Ax	125.000	cm2	Cross-section area		Eorces
	Detailed	Ay	78.511	cm2	Shear area - Y-axis		Detailed
Class of section = 1	Logico	Az	55.655	cm2	Shear area - Z-axis		Decidien
LATERAL BUCKLING		bx .	121.000	cm4	Torsional constant		
XLT = 1.00		ly	45730.000	cm4	Moment of inertia of a section about the Y-axis		
		lz	2347.000	cm4	Moment of inertia of a section about the Z-axis		
BUCKLING Z	Calc. Note	Wpły	2234.000	cm3	Plastic section modulus about the Y (major) axis		Calc. Note
\mathbf{X}		Wplz	379.000	cm3	Plastic section modulus about the Z (minor) axis		
	Parameters	h	46.74	cm	Height of cross-section		Parameters
		b	19.28	cm	Width of cross-section		
	Help	tf	1.96	cm	Flange thickness		Help
SECTION CHECK		tw	1.14	cm	Web thickness		
My,Ed/My,N,Rd = 1.00 < 1.00 (6.2.9.1.(2))		ry	19.13	cm	Radius of gyration - Y-axis		
		rz	4.33	cm	Radius of gyration - Z-axis		
MEMBER STABILITY CHECK					Material:		
Not analyzed		Name			\$235 (\$235)	7	
		fy	235.00	MPa	Design yield strength of material (3.2)	-	
]



AUTODESK

STEEL DESIGN

CODE: EN 1993-1:2005, Eurocode 3: Design of steel structures. ANALYSIS TYPE: Member Verification										
CODE GROUP: MEMBER: 1 Bar 1 0.00 m	POINT: 1		COORDINATE: $x = 0.00 L =$							
LOADS: Governing Load Case:	1 STA1									
MATERIAL: S235 (S235) fy = 2	35.00 MPa									
SECTION PAR h=46.74 cm b=19.28 cm tw=1.14 cm tf=1.96 cm	RAMETERS: UB 457x191x gM0=1.00 Ay=78.511 cm2 Iy=45730.000 cm4 Wply=2234.000 cm3	98 gM1=1.00 Az=55.655 cm2 Iz=2347.000 cm4 Wplz=379.000 cm3	Ax=125.000 cm2 Ix=121.000 cm4							
INTERNAL FORCES A N,Ed = 1400.0 kN Nc,Rd = 2937.5 kN Nb,Rd = 2937.5 kN	AND CAPACITIES: My,Ed = -342.2 kN*m My,pl,Rd = 525.0 kN*m My,c,Rd = 525.0 kN*m My,N,Rd = 342.5 kN*m		Class of section = 1							
	CKLING PARAMETERS:									
BUCKLING PARAME About Y axis:	TERS:	About Z axis	5:							
VERIFICATION FORM Section strength check: N,Ed/Nc,Rd = 0.48 < 1.0 My,Ed/My,c,Rd = 0.65 < My,Ed/My,N,Rd = 1.00	00 (6.2.4.(1)) < 1.00 (6.2.5.(1))									

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

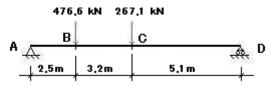
Example taken from Designer's Guide to EN 1993-1-1 L.Gardner and D.A.Nethercot, Thomas Telford Publishing, 2005

TITLE:

Lateral torsional buckling resistance (Example 6.8 page 74).

SPECIFICATION:

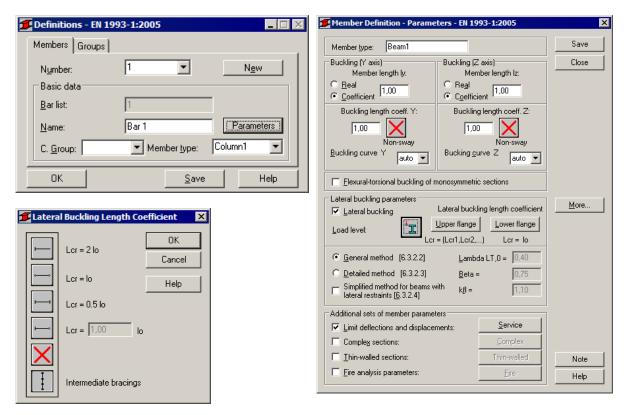
Simply supported primary beam supports two secondary beams, represented with the concentrated load as shown below. The secondary beams create full lateral restraint of the primary beam web at these points. Section UB 762x267x173 is checked in grade S275 steel. The loads given are at the ultimate limit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

15



🗲 Internal bracings	×
	10.80 m
Buckling Y Buckling Z Lateral buckling-upper flange Lateral buckling	g-lower flange
Coordinates of the existing bracings	Automatic detection of bracings
Define manually coordinates of the existing bracings	Add bracings at points where adjoining elements occur
2,50; 5,70 m	Add bracings at points where bending moments equal
	zero
Basic scheme of a member	Bracing detection preview For member no.:
	For load case:
Buckling coefficients of component segments	m
1,00; 1,00; 1,00	
Cancel	Help

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 [and other products] are registered trademarks or trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and/or other countries. All other brand names, product names, or trademarks belong to their respective holders. Autodesk reserves the right to alter product and services offerings, and specifications and pricing at any time without notice, and is not responsible for typographical or graphical errors that may appear in this document. © 2024 Autodesk, Inc. All rights reserved.



🗲 Calculations - EN 1993-1:2005	Configuration
Verification options	Calculation points
<u>M</u> ember verification:	Number of points:
O Code group verification:	Characteristic points
C Code group design:	Calculation parameters
Optimization Options	Efficiency ratio: 1,00 Help
Loads Limit state	Maximum slenderness: 210,00
Cases: 1 List Vitimate	Components of complex bars are
	not taken into account
Save calculation results	□ Shear verification in elastic state [6.2.6]
OK Configuration Calculations Help	and the set of the set

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.

1	EN 1993-1:2005 - Member Verification (ULS) 1									
	Results Messages									
	Member	Section	Material	Lay	Laz	Ratio	Case			Help
	1 Bar 1	UB 762x267x1	S275	35.35	193.55	0.91	1 STA1		r Ratio	
									Analysis	Map
									Calculation po Division:	n = 101
1										none
l									Additional:	none

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.

FRESULTS - Code - EN 1993-1:2005	_ 🗆 🗙	🛙 🎏 RESULTS - Code - EN 1993-1:2005	_ 🗆 🗙
Image: Section 0K Bar: 1 Bari Image: Section 0K Image: Section 0K Image: Section 0K Image: Section 0K Image: Section 0K Image: Section 0K Image: Section 0K Image: Section 0K Image: Section 0K Image: Section 0K Image: Section 0K Image: Section 0K	<u>(СОК</u>)	Bar: 1 Bar: </td <td>OK)</td>	OK)
Simplified results Detailed results	Change	Simplified results Detailed results	Change
FORCES My, Ed = 1276, 7 kN*m My, p.R.H = 1703, 6 kN*m		Symbol Values Unit Symbol description Section Cross-section properties: UB 762x67x173	
My.c.Rd = 1703.6 kN*m Vz.Ed = -251.5 kN			
Vz.c.Rd = 1825.9 kN Mb.Rd = 1401.2 kN*m	Eorces	Ax 220.000 cm2 Cross-section area	Eorces
M0,hd = 1401.2 KN m Class of section = 1	Det <u>a</u> iled	Ay 121.931 cm2 Shear area - Y-axis Az 115.002 cm2 Shear area - Z-axis	Detailed
LATERAL BUCKLING		Az 115.002 cm2 Shear area - 2-axis tx 267.000 cm4 Torsional constant	
		V 205300.000 cm4 Moment of inertia of a section about the Y-axis	
Image: style="text-align: center;">z = 1.00 Mcr = 4311.9 kN/m Curve,LT - b XLT = 0.82 Image: style="text-align: center;">Lcr,upp=5.10 m Lam LT = 0.63 fiLT = 0.77		Iz 6850.000 cm4 Moment of inertia of a section about the 7-axis	
	I	Wply 6195.000 cm3 Plastic section modulus about the Y (major) axis	
BUCKLING Z	Calc. Note	Wp/y Orestor Cristic section Modulate data the Friday of the Friday	Calc. Note
X	Parameters	h 76.20 cm Height of cross-section	Parameters
		b 26.67 cm Width of cross-section	
		tf 2.16 cm Flange thickness	
	Help	tw 1.43 cm Web thickness	Help
SECTION CHECK Mv.Ed/Mv.c.Rd = 0.75 < 1.00 (6.2.5.(1))		ry 30.55 cm Radius of gyration - Y-axis	
Vz,Ed/Vz,c,Rd = 0.14 < 1.00 (6.2.6.(1))		rz 5.58 cm Radius of gyration - Z-axis	
MEMBER STABILITY CHECK		Material:	
My,Ed/Mb,Rd = 0.91 < 1.00 (6.3.2.1.(1))		Name S275 (S275)	
		fy 275.00 MPa Design yield strength of material (3.2)	
]		

Autodesk [and other products] are registered trademarks or trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and/or other countries. All other brand names, product names, or trademarks belong to their respective holders. Autodesk reserves the right to alter product and services offerings, and specifications and pricing at any time without notice, and is not responsible for typographical or graphical errors that may appear in this document. © 2024 Autodesk, Inc. All rights reserved.

AUTODESK

STEEL DESIGN

CODE: EN 1993-1:2005, Eurocode 3: Design of steel structures. ANALYSIS TYPE: Member Verification						
CODE GROUP: MEMBER: 1 Bar 1 5.72 m	POINT: 54	coc	DRDINATE: x = 0.53 L =			
LOADS: Governing Load Case: 1	STA1					
MATERIAL: S275 (S275) fy = 27	5.00 MPa					
	AMETERS: UB 762x267x	173				
h=76.20 cm	gM0=1.00	gM1=1.00				
b=26.67 cm	5	Az=115.002 cm2	Ax=220.000 cm2			
tw=1.43 cm	Iy=205300.000 cm4		Ix=267.000 cm4			
tf=2.16 cm	Wply=6195.000 cm3	Wplz=807.000 cm3				
INTERNAL FORCES A						
	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			
	$My,c,Ku = 1703.0 KN \cdot III$		$V_{z,c,Rd} = -231.5 \text{ kN}$ Vz,c,Rd = 1825.9 kN			
	Mb,Rd = 1401.2 kN*m		V 2, C, Ru = 1023.9 KiV			
	100, Rd = 1101.2 kiv m		Class of section $= 1$			
	BUCKLING PARAMETER	S:				
z = 1.00	Mcr = 4311.9 kN*m	,	XLT = 0.82			
Lcr,upp=5.10 m	$Lam_LT = 0.63$	fi,LT = 0.77				
	ERS:					
About Y axis:		About Z axis:				
VERIFICATION FORM Section strength check: My,Ed/My,c,Rd = 0.75 < Vz,Ed/Vz,c,Rd = 0.14 < 1 Global stability check of a My,Ed/Mb,Rd = 0.91 < 1.	1.00 (6.2.5.(1)) .00 (6.2.6.(1)) <i>nember:</i>					
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 XLT	0,82	0,82





VERIFICATION EXAMPLE 5 - Combined bi-axial bending and compression

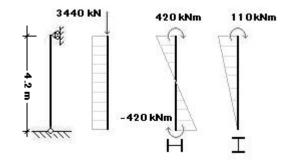
Example taken from Designer's Guide to EN 1993-1-1 L.Gardner and D.A.Nethercot, Thomas Telford Publishing, 2005

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.

🗲 Definitions - EN 1993-1:2005	🗲 Buckling Diagrams	×
Members Groups Number: 1 Basic data Bar list: 1 Name: Bar 1 Parameters: C. Group: OK Save Help	$ \begin{array}{c c} \hline \\ 1 \\ 1 \\ 1 \\ 1 \\ 1 \\ 0.5 \\ 0.7 \\ 2.0 \\ 0.7 \\ 2.0 \\ 0.7 \\ 2.0 \\ 0.7 \\ 2.0 \\ 0.7 \\ 2.0 \\ 0.7 \\ 2.0 \\ 0.7 \\ 2.0 \\ 0.7 \\ 0.$	OK Cancel Help

Set Lateral buckling checkbox.

AUTODESK

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



Member Definition - Parame	tors - FN 10	03-1-2005	×
Frember Dennition - Parame	ters - LN 15	55-1-2005	<u> </u>
Member type: Column1			Save
Buckling (Yaxis) Member length ly:	– Buckling (Z Men	' axis) nber length lz:	Close
C <u>R</u> eal ⊙ <u>C</u> oefficient 1,00	C Re <u>a</u> l © C <u>o</u> effici	ent 1,00	
Buckling length coeff. Y:	Buckling	g length coeff. Z:	
0,70	1,00	1.0	
Non-sway Buckling curve Y	Bucking <u>c</u> u	Non-sway urve Z auto 💌	
Elexural-torsional buckling of r	nonosymmetri	c sections	
Lateral buckling parameters			
Lateral buckling	Lateral buckl	ing length coefficient	<u>M</u> ore
Load level:	<u>U</u> pper flange	Lower flange	
	Lcr = lo	Lcr = lo	
• <u>G</u> eneral method [6.3.2.2]	Lambo	da LT,0 = 0,40	
© <u>D</u> etailed method [6.3.2.3]	<u>B</u> eta =	0,75	
□ Simplified method for beams w lateral restraints [<u>6</u> .3.2.4]	vith k <u>f</u> l =	1,10	
Additional sets of member paramet	ters		
Limit deflections and displace	ments:	<u>S</u> ervice	
Complex sections:		<u>C</u> omplex	
<u>I</u> hin-walled sections:		Thin-walled	Note
Eire analysis parameters:		Eire	Help

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.

🗲 Calculations - EN 1993-1	:2005
Verification options	
Member verification:	1 List
Code group verification:	List
Code group <u>d</u> esign:	List
<u>Optimization</u>	Options
Loads	Limit state
Cases: 1	ist ↓ Itimate
Calculation archive	List Erviceability
OK Configu	uration Calculations Help

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.





5 EN 1993-1:200	5 - Member Verifi	ation (ULS)	1					
Results Message	es						Calc. Note	<u>C</u> lose
Member	Section	Material	Lay	Laz	Ratio	Case		Help
1 Bar 1	UC 305x305x2	S 275	20.30	51.55	0.97	1 STA1	n Ratio	
							Analysis	Мар
							Calculation poir Division: n	nts = 3
								one

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.

#RESULTS - Code - EN 1993-1:2005		🗲 RESULTS - Co	de - EN 1993-1:	2005			_ 🗆 🗙
▲uto Bar 1 Bar 1 UC 305:375:240 ▼ Section 0K ● Load case 1 STA1 ●	<u> </u>	UC 305x305x24	<u>Auto</u>	Bar: Point / Coo Load case:			<u>(СОК</u>)
Simplified results Dotated results	Change	Simplified results	Detailed results	1			Change
FORCES N.E.d = 3440 kN My.E.d = -420 kN*m Mz.E.d = 110 kN*m Vy.E.d = -26 kN		Symbol	Values	Unit	Symbol description Section	-	1
Nc,Rd = 8415 kN My,plRd = 1167 kN*m Mz,plRd = 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				Cross-	section properties: UC 305x305x240		
My,N,Rd = 774 kN*m Mz,N,Rd = 503 kN*m Vz,c,Rd = 1372 kN	Eorces	Ax	306.000	cm2	Cross-section area		Eorces
Mb,Rd = 1150 kN*m	Detailed	Ay	249.236	cm2	Shear area - Y-axis		Detailed
Class of section = 1		Az	86.435	cm2	Shear area - Z-axis		
LATERAL BUCKLING		bx	1271.000	cm4	Torsional constant		
		ly	64200.000	cm4	Moment of inertia of a section about the Y-axis		
Lor,low=4.20 m Lam_LT = 0.26 fi,LT = 0.54		lz	20310.000	cm4	Moment of inertia of a section about the Z-axis		
BUCKLING Y BUCKLING Z	Calc. Note	Wply	4243.000	cm3	Plastic section modulus about the Y (major) axis		Calc. Note
Ly = 4.20 m Lam_y = 0.23 Lz = 4.20 m Lam_z = 0.59		Wplz	1945.000	cm3	Plastic section modulus about the Z (minor) axis		
07 Lcr.y = 2.94 m Xy = 0.99 10 Lcr.z = 4.20 m Xz = 0.79	Parameters	h	35.26	cm	Height of cross-section		Parameters
Lamy = 20.30 kzy = 0.79 Lamz = 51.55 kzz = 0.78		b	31.79	cm	Width of cross-section		
	Help	tf	3.77	cm	Flange thickness		Help
	Help	tw	2.30	cm	Web thickness		neip
Mv.Ed/Mv.c.Rd + Mz.Ed/Mz.c.Rd = 0.57 < 1.00 (6.2.5(1))		ry	14.48	cm	Radius of gyration - Y-axis		
VzEd/Vz,c,Rd = 0.15 < 1.00 [6.2.6.[1]]		rz	8.15	cm	Radius of gyration - Z-axis		
MEMBER STABILITY CHECK Lamv = 20.30 < Lam.max = 210.00 Lamz = 51.55 < Lam.max = 210.00 STABLE					Material:		
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))		Name			\$ 275 (\$275)		
		fy	275.00	MPa	Design yield strength of material (3.2)	1	

STEEL DESIGN

CODE GROUP: MEMBER: 1 Bar 1	POINT: 3	COORD	DINATE: $x = 1.00 L = 4.20 n$
LOADS: Governing Load Case:	1 STA1		
MATERIAL: S 275 (S275) fy =	= 275.00 MPa		
SECTION PA	ARAMETERS: UC 305x305x	<240	
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 tf=3.77 cm	Iy=64200.000 cm4 Wply=4243.000 cm3	Iz=20310.000 cm4 Wplz=1945.000 cm3	Ix=1271.000 cm4
INTERNAL FORCES	AND CAPACITIES:		
N,Ed = 3440 kN	My,Ed = -420 kN*m	Mz,Ed = 110 kN*m	
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		

21

AUTODESK

LATERAL B z = 0.00 Lcr,low=4.20 m	UCKLING PARAMETER Mcr = 16778 kN*m Lam_LT = 0.26	S: Curve,LT - a fi,LT = 0.54	XLT = 0.99			
BUCKLING PARAMETE $1 \\ 0.7$ About Y axis Ly = 4.20 m Lcr, y = 2.94 m	5: Lam_y = 0.23 Xy = 0.99	Lz = 4.20 m Lcr, z = 4.20 m	$Lam_z = 0.59$ 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 + Vy,Ed/Vy,c,Rd = 0.01 < 1. Vz,Ed/Vz,c,Rd = 0.15 < 1. <i>Global stability check of m</i> Lambda,y = 20.30 < Lamb	00 (6.2.6.(1)) nember:	0.34 < 1.00 (6.2.9.1.(6)) oda, z = 51.55 < Lambda, max	= 210.00 STABLE			
My,Ed/Mb,Rd = 0.37 < 1.0 N,Ed/(Xy*N,Rk/gM1) + ky	0 (6.3.2.1.(1)) yy*My,Ed/(XLT*My,Rk/gN	11) + kyz*Mz,Ed/(Mz,Rk/gN	$\begin{array}{l} (A1) = 0.66 < 1.00 (6.3.3.(4)) \\ (A1) = 0.97 < 1.00 (6.3.3.(4)) \\ (A1) = 0.97 < 1.00 (A1) \\ (A1) = 0.97 < 0.00 (A1) \\ (A1) = 0.00 (A1) (A1) (A1) \\ (A1) = 0.00 (A1) (A$			

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

23



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	
$\alpha_{cc} = 0.85$		
Steel:	fyk=355	[MPa]

LOADS:

Bending moment M = 100kNm [cm²]

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 $\alpha_{cc} = 0.85$. The default value for the general edition of the code is $\alpha_{cc} = 1.0$. 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*).



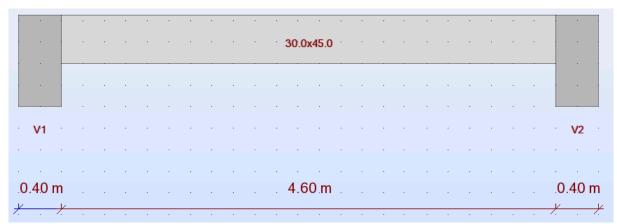
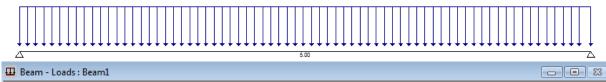


Fig. 1.1 Beam geometry



-													
		Case number	Distributed load	Nature	Li st	Positi on	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

Job Preferences				? 💌
DEFAL	ILTS	Database Name	Database Description	•
Vehicle loads Standard loads Building soils Bolts Anchor bolts Reinforcing bars Wire fabrics Design codes	EN 1992-1-1 EC2 - ICEL EC2 - ITALI NEN-EN PN 2002_#			•
Open default par Save current paramet			Cancel	Help

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



AUTODESK

Job Preferences			٢.
🚅 🖶 🗙 🔆 👘	DEFAULTS	•	
 Units and Formats Materials Databases Design codes Loads Structure Analysis Modal Analysis Non-linear Analysis Seismic Analysis Work Parameters 	<u>S</u> teel/Aluminum structures: St <u>e</u> el connections: <u>T</u> imber structures: <u>B</u> C structures: <u>G</u> eotechnical:	EN 1993-1:2005 EN 1993-1-8:2005 ENV 1995-1-1:1992 EN 1992-1-1:2004 AC:2008 ENV 1997-1:1994 <u>More codes</u>	
_	arameters as default	OK Cancel Help	

Partial Factors for a Code EN 1992-1	-1:20	04 AC:2008			? 💌		
RC structures							
EN 1992-1-1:2004		Coefficient	Value	Code reference	•		
	8	k1 (redistribution)	0.44	1992-1-1 5.5 (4)			
SFS-EN 1992-1-1	9	k2 (redistribution)	Auto	1992-1-1 5.5 (4)			
UNI-EN1992-1-1	10	k3 (redistribution)	0.54	1992-1-1 5.5 (4)			
PN-EN 1992-1-1:2008	11	k4 (redistribution)	Auto	1992-1-1 5.5 (4)	=		
FN-EN 1332-1-1.2006	12	k5 (redistribution)	0.70	1992-1-1 5.5 (4)			
→ User-defined	13	k6 (redistribution)	0.80	1992-1-1 5.5 (4)			
	14	α _{cc}	0.85	1992-1-1 3.1.6 (1)P			
	15	α _{ct}	1.00	1992-1-1 3.1.6 (2)P			
	16	$\epsilon_{ud} / \epsilon_{uk}$	0.90	1992-1-1 3.2.7 (2)	-		
Copy to user's set							
OK Cancel Help							

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.



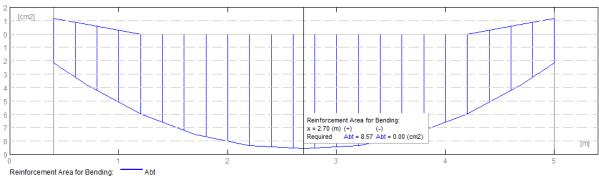


Fig. 1.5. Theoretical (required) areas of reinforcement in beam.

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 $\alpha_{cc} = 0.85$. The default value for the general edition of the code is $\alpha_{cc} = 1.0$. 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	γc	γ_{s}	α_{cc}	bottom reinf.
				A _{s1} -Robot results
Handbook example	1.5	1.15	0.85	8.57 cm^2
(general Eurocode 2 edition				
with modified α_{cc}				
EN 1992-1-1:2004	1.5	1.15	1.0	8.45 cm^2
AC:2008				
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	1.5	1.15	0.85	8.57 cm^2
NA2005				
NS-EN 1992-1-	1.5	1.15	0.85	8.57 cm^2
1:2004/NA:2008				
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:

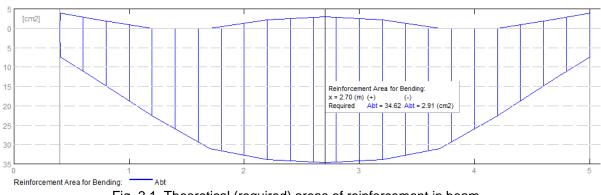
Calculate the reinforcement in rectangular section at simple bending at ULS. In this example, the results of the program are compared against [1]. The data is the same as in Verification problem 1, except for the bending moment which is equal to M=320 kNm.

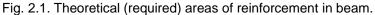
RESULTS OF LONGITUDINAL REINFORCEMENT (REINFORCEMENT FOR BENDING) CALCULATION:

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_{sI}	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.





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 $\alpha_{cc} = 0.85$. The default value for the general edition of the code is $\alpha_{cc} = 1.0$.

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.

29

AUTODESK

Code	γc	γ_{s}	α_{cc}	bottom reinf. <i>A</i> _{s1} - Robot results	top reinf. A_{s2} -Robot results
Handbook example	1.5	1.15	0.85	34.73 cm ²	2.92 cm^2
(general Eurocode 2 edition with					
modified α_{cc}					
EN 1992-1-1:2004	1.5	1.15	1.0	34.27 cm ²	$0.0 \ {\rm cm}^2$
AC:2008					
PN-EN 1992-1-	1.4	1.15	1.0	33.09 cm^2	$0.0 \ {\rm cm}^2$
1:2008					
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	1.45	1.2	1.0	35.12 cm^2	0.0 cm^2
NA:2007					
BS EN1992-1-	1.5	1.15	0.85	34.73 cm^2	2.92 cm^2
1:2004 NA2005					
NS-EN 1992-1-	1.5	1.15	0.85	34.73 cm ²	2.92 cm^2
1:2004/NA:2008					
NF EN 1992-1-	1.5	1.15	1.0	34.27 cm ²	$0.0 \ {\rm cm}^2$
1/NA:2007					

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:

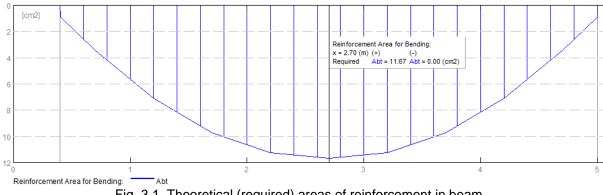
Calculate the reinforcement in rectangular section at bending with compression at ULS. In this example, the results of the program are compared against [1]. The data is the same as in Verification problem 1, except of the forces which are: bending moment M=150 kNm, and compressive force N=150 kNm.

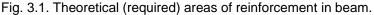
RESULTS OF LONGITUDINAL REINFORCEMENT (REINFORCEMENT FOR BENDING) CALCULATION:

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_{sl}	11.62 cm^2	11.67 cm ²

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





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 $\alpha_{cc} = 0.85$. The default value for the general edition of the code is $\alpha_{cc} = 1.0$. 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	γc	γ_{s}	α_{cc}	bottom reinf. A _{s1} -Robot results
Handbook example	1.5	1.15	0.85	11.67 cm ²
(general Eurocode 2 edition				
with modified α_{cc}				
EN 1992-1-1:2004	1.5	1.15	1.0	11.17 cm^2
AC:2008				
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	1.5	1.15	0.85	11.67 cm ²
NA2005				
NS-EN 1992-1-	1.5	1.15	0.85	11.67 cm ²
1:2004/NA:2008				
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:

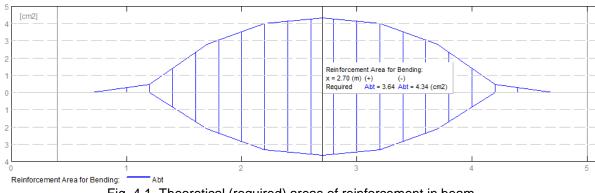
Calculate the reinforcement in rectangular section at bending with compression at ULS. In this example, the results of the program are compared against [1]. The data is the same as in Verification problem 1, except of the forces which are: bending moment M=150 kNm, and compressive force N=1000 kNm.

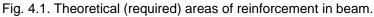
RESULTS OF LONGITUDINAL REINFORCEMENT (REINFORCEMENT FOR BENDING) CALCULATION:

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_{sl}	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.





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 $\alpha_{cc} = 0.85$. The default value for the general edition of the code is $\alpha_{cc} = 1.0$. 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	γc	γ_{s}	α_{cc}	bottom reinf. A_{sl} -	top reinf. A_{s2} -
				Robot results	Robot results
Handbook example	1.5	1.15	0.85	3.64 cm^2	4.34 cm^2
(general Eurocode 2					
edition with modified					
$lpha_{ ext{cc}}$					
EN 1992-1-1:2004	1.5	1.15	1.0	4.75 cm^2	$0.0 \ {\rm cm}^2$
AC:2008					
PN-EN 1992-1-1:2008	1.4	1.15	1.0	3.24 cm^2	$0.0 \ {\rm cm}^2$
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	1.45	1.2	1.0	4.13 cm^2	$0.0 \ {\rm cm}^2$
NA:2007					
BS EN1992-1-1:2004	1.5	1.15	0.85	3.64 cm^2	4.34 cm^2
NA2005					
NS-EN 1992-1-	1.5	1.15	0.85	3.64 cm^2	4.34 cm^2
1:2004/NA:2008					
NF EN 1992-1-	1.5	1.15	1.0	4.75 cm ²	0.0 cm^2
1/NA:2007					

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:	$q_D = 30 [\text{kN/m}]$					
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.

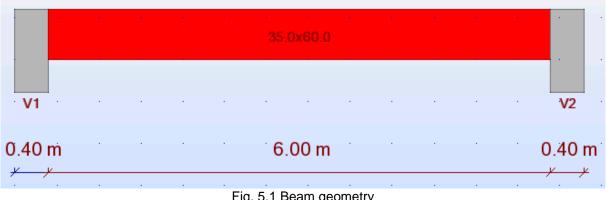


Fig. 5.1 Beam geometry

AUTODESK



🖽 Beam - Loads : Beam1												
	Case number	Distributed load	Nature	Li st	Positi on	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:

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

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:

$$\begin{split} V_{Rd,c} &= \left[C_{Rd,c} k (100 \rho_l f_{ck})^{1/3} + k_1 \sigma_{cp} \right] b_w d = 103.69 k N \\ C_{Rd,c} &= 0.18 / \gamma_c = 0.12 \\ k &= 1 + \sqrt{200/d} = 1.61 \leq 2.0 \\ d &= 600 - 65 = 535 mm \end{split}$$
(position of bottom bars is averaged for two layers)
$$\rho_l &= \frac{A_{sl}}{b_w d} = 0.0117 \\ A_{sl} &= 2199 mm^2 \\ b_w &= 350 mm \\ f_{ck} &= 20 MPa \\ \\ \text{But should not be smaller than:} \\ V_{Rd,c} &= \left[V_{\min} + k_1 \sigma_{cp} \right] b_w d = 59.9 k N \\ v_{\min} &= 0.035 k^{3/2} f_{ck}^{1/2} = 0.32 \end{split}$$

36



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 midspan) (*Fig.5.3*).

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

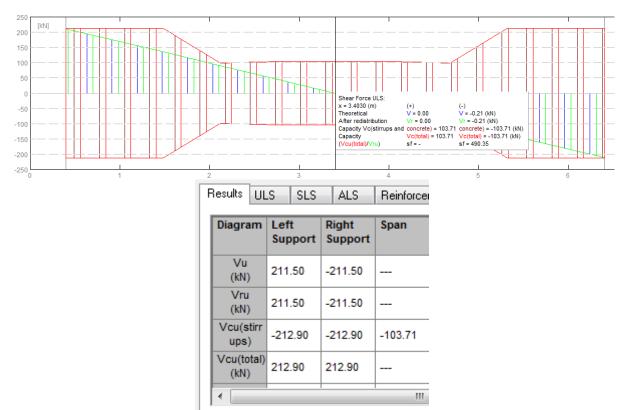


Fig. 5.3 Shear force distribution and shear capacity

• CALCULATION OF SHEAR CAPACITY OF A BEAM WITH SHEAR REINFORCEMENT:

Since, at the support face $V \ge V_{Rd,c}$ 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 $V_{\rm Rd,s} = V_{\rm x=0.4}$, the required spacing of stirrups near the support is:

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

$$A_{sw} = 0.000101m^{2}$$

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

$$z = 0.9d = 0.49m$$

$$d = 0.6 - 0.059 = 0.541m$$
(for bottom b

(for bottom bars at the support)

Autodesk [and other products] are registered trademarks or trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and/or other countries. All other brand names, product names, or trademarks belong to their respective holders. Autodesk reserves the right to alter product and services offerings, and specifications and pricing at any time without notice, and is not responsible for typographical or graphical errors that may appear in this document. © 2024 Autodesk, Inc. All rights reserved.



$$f_{ywd} = f_{ywk} / \gamma_s = 434.8MPa$$
$$f_{ywk} = 500MPa$$
$$\gamma_s = 1.15$$

 $\cot \theta = 1.0$

(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 = $\frac{\alpha_{cw}b_w zv_1 f_{cd}}{\cot \theta + \tan \theta} = 627.4kN$

$$\alpha_{cw} = 1.0$$

$$\nu_1 = 0.552$$

$$f_{cd} = f_{ck} / \gamma_c = 13.33MPa$$

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		

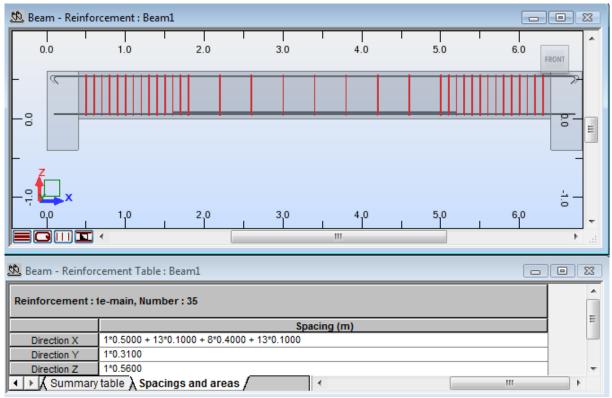


Fig. 5.4 Shear reinforcement distribution (see Direction X in the Reinforcement table)

Autodesk [and other products] are registered trademarks or trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and/or other countries. All other brand names, product names, or trademarks belong to their respective holders. Autodesk reserves the right to alter product and services offerings, and specifications and pricing at any time without notice, and is not responsible for typographical or graphical errors that may appear in this document. © 2024 Autodesk, Inc. All rights reserved.



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	γc	γ_{s}	α_{cc}	Shear capacity $V_{Rd,c}$	Shear capacity
				Ku,c	$V_{Rd,s}$
EN 1992-1-1:2004	1.5	1.15	1.0	103.71 kN	212.9 kN
AC:2008					
(manual calculation)					
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	1.5	1.15	0.85	103.71 kN	236.03 kN
NA2005					
NS-EN 1992-1-	1.5	1.15	0.85	103.71*	236.03 kN
1:2004/NA:2008					
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: 5\u00e920

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





$\downarrow \downarrow $		
Δ	7.50	Δ

	Case number	Distributed load	Nature	Li st		Coord. syste			Coordinates	Projection	x1 (m)	p1 (kN/m)
	DL1	uniform	dead loa	1	upp	Local	Z	1.35	absolute	Not projected		22.76
ľ		1	-							1	1	

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*).

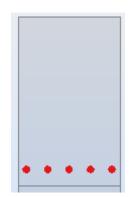


Fig. 5.2 Reinforcement ($5\phi 20$) assumed in [1]

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



42

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.

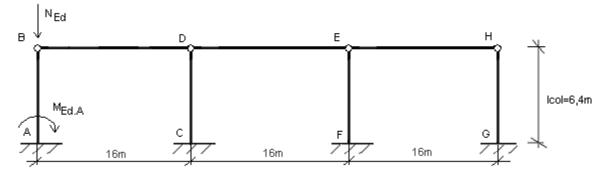


Fig. 1.1. The model of the frame with the analyzed column AB.

GEOMETRY:

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

MATERIAL:

Concrete:	C30/37	
$\alpha_{cc} = 0.85$		
Creep coeffi	cient: 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 ϕ 20 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).

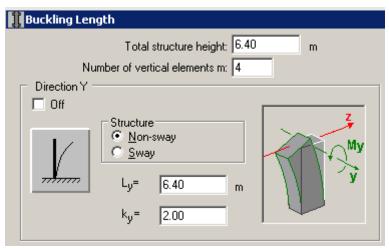


Fig. 1.2 Buckling parameters

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)*.

Hay Job Preferences				? 💌
🖙 🖶 🗙 🔆 DEFAU	LTS			•
 	• •			
 Steel and timber s Vehicle loads Standard loads Building soils Bolts Anchor bolts Reinforcing bars Wire fabrics Design codes 	Database EN 1992-1-1 EC2 - ICEL EC2 - ITALI NEN-EN PN 2002_#	Database Name	Database Description	4
🔍 🔤 🔍 Open default par	ameters			
Save current paramet	ers as default	ОК	Cancel	Help

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

Autodesk [and other products] are registered trademarks or trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and/or other countries. All other brand names, product names, or trademarks belong to their respective holders. Autodesk reserves the right to alter product and services offerings, and specifications and pricing at any time without notice, and is not responsible for typographical or graphical errors that may appear in this document. © 2024 Autodesk, Inc. All rights reserved.

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

Simplified second order analysis method
• Nominal stiffness
O Nominal curvature

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*).

Design for simple bending	
My direction	O Mz direction

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*).

Corner bars
Diameter: 20 💌
Maximal number of bars in a bundle
1 💌
Intermediate bars
Identical diameters
Diameter: 20 💌
Maximal number of bars in a bundle
1 💌

Fig. 1.7 Parameters of reinforcement

The authors of [1] use the partial factor $\alpha_{cc} = 0.85$. The default value for the general edition of the code is $\alpha_{cc} = 1.0$. 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).*



46

Job Preferences			? 🔀	
😅 层 🗙 🔆 🛛 🖻	EFAULTS		-	
 Units and Formats Materials Databases Design codes Loads Structure Analysis Modal Analysis Non-linear Analysis Seismic Analysis Work Parameters 	<u>S</u> teel/Aluminum structures: St <u>e</u> el connections: <u>T</u> imber structures: <u>R</u> C structures: <u>G</u> eotechnical:	EN 1993-1:2005 EN 1993-1-8:2005 ENV 1995-1-1:1992 EN 1992-1-1:2004 AC:2008 ENV 1997-1:1994 <u>More codes</u>	• • • •	
😪 Open default parameters				
Save current parameters as default OK Cancel Help				
Rin Partial Factors for a Code EN 1	992-1-1:2004 AC:2008		? ×	

EN 1992-1-1:2004		Coefficient	Value	Code reference
	8	k1 (redistribution)	0.44	1992-1-1 5.5 (4)
SFS-EN 1992-1-1	9	k2 (redistribution)	Auto	1992-1-1 5.5 (4)
W UNI-EN1992-1-1	10	k3 (redistribution)	0.54	1992-1-1 5.5 (4)
THE THE 1002 1 1-2000	11	k4 (redistribution)	Auto	1992-1-1 5.5 (4)
PN-EN 1992-1-1:2008	12	k5 (redistribution)	0.70	1992-1-1 5.5 (4)
User-defined	13	k6 (redistribution)	0.80	1992-1-1 5.5 (4)
	14	αcc	0.85	1992-1-1 3.1.6 (1)P
	15	α _{et}	1.00	1992-1-1 3.1.6 (2)P
	16	$\epsilon_{ud} / \epsilon_{uk}$	0.90	1992-1-1 3.2.7 (2)
			Copy to user's set	

Fig. 1.8 Definition of partial factors

NOTE: The program automatically generates smaller reinforcement (8 ϕ 20 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*)

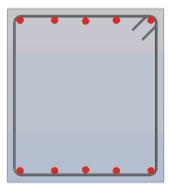


Fig. 1.9 Reinforcement assumed for the calculation (10 ϕ 20).

	(Unit)	[1]	Robot (results presented in calculation note)
$\lambda_{ m lim}$	(-)	32.2	32.3
$lpha_{_h}$	(-)	0.791	0.791
$\alpha_{_m}$	(-)	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
с	(-)	10	10
e_2	(cm)	13.7 (14.1)**	14.0
M_{Ed}	(kNm)	289.8 (293.9)**	291.97

RESULTS OF BUCKLING ANALYSIS - NOMINAL CURVATURE METHOD:

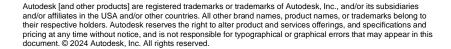
As can be seen, a very good agreement concerning the final results is obtained, even if some small discrepancies may occur in partial results.

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 $f_{yd} = 350 MPa$ value, while program uses $f_{yd} = f_{yk} / \gamma_s = 357 MPa$

** - 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.



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) por (3) can be applied). It

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	γc	α_{cc}	Design moment M_{Ed}
EN 1992-1-1:2004	1.5	1.0	317.38 kN
AC:2008			
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	1.45	1.0	318.57 kN
NA:2007			
BS EN1992-1-1:2004	1.5	0.85	311.39 kN
NA2005			
NS-EN 1992-1-	1.5	0.85	311.39 kN
1:2004/NA:2008			
NF EN 1992-1-	1.5	1.0	317.38 kN
1/NA:2007			

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 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.



Mew Thickness
Homogeneous Orthotropic
h
ţ.
Label: h240 Color: Auto 🗸
Constant Th = 24.0 (cm)
○ Variable along a line
◯ Variable on a plane
Point coordinates Thicknesses
(m) (cm) P1: 7.0000; 2.0000; 0.0000 0.0
P2: 7.0000; 2.0000; 0.0000 0.0
P3: 7.0000; 2.0000; 0.0000 0.0
Reduction of the 1.00 Mg
moment of inertia
Parameters of foundation elasticity
Material: C20/25
Add Close Help

Fig. 1.1 Slab thickness

Material Definition Steel Concrete Aluminum Tim	ber Other				
<u>N</u> ame: fck15					
Elasticity Young modulus, <u>E</u> : Poisson ratio, <u>v</u> : Shear modulus, <u>G</u> :	27000.00 (MPa) Resistance 0.2 Characteristic 15.00 11250.00 (MPa) Sample:				
Force <u>d</u> ensity (unit weight): <u>T</u> hermal expansion coefficient: Da <u>m</u> ping ratio:	24.53 (kN/m3) 0.000010 (1/°C) 0.04				
Add Delete OK Cancel Help					

Fig. 1.2 Definition of new concrete type

FEN 1992-1-1:2004 AC:2008 Reinforcement Par					
General Materials SLS Parameters Reinforcement					
Bar dimensions					
d1: 12 ▼ d2: 12 ▼					
d1': 12 ▼ d2': 12 ▼					
Cover (cm)					
c1: 1.9 c2: 1.9					
c1': 0.0 c2': 0.0					
Deviations					
Unidirectional reinforcement					
Membrane reinforcement in one layer					
Minimum reinforcement					
© None					
For FE for which reinforcement As>0					
For the whole panel					
Note Add Close Help					

Fig. 1.3 Definition of covers of reinforcement

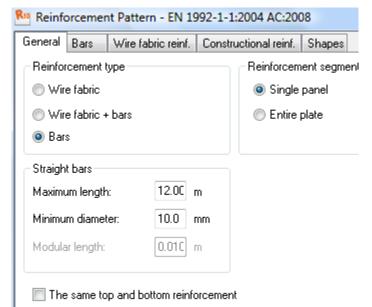


Fig. 1.4 Selection of reinforcement with bars



Reinforcement Pattern - EN 1992-1-1:2004 AC:2008						
structional reinf. Shapes						
Top reinforcemen Diameter Direction X Direction Y	12 • 12 •					
Spacing (cm) © Direction X © Direction Y	7.0					
Preferred reinforcement spacing Direction X Maximum 40.0 cm Minimum 3.0 cm						
Minimum	3.0 cm					
	Structional reinf. Shapes Top reinforcemen Diameter Direction X Direction Y Spacing (cm) Direction X Direction Y					

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.





1 11	Plate ar	nd Shell R	einforcer	ment			
F	Punching						
h	Verifical	tion points			Point grouping		
	Name:	S1 :	30x30		List:	-	
				>>	New	Delete	
	Ne	w	Delete		Maximum punching for	ce (kN)	
IL	Туре:	inter	mal	•	160.00		
	Position	n (m)					
	x = 2.0000 y = 2.0000 Node number: 2						
	Punching: from bottom 👻						
	Head Dimensions (cm)						
	Ty			67 H	a = 30.0		
	۲	rectangula	ſ		b = 30.0		
	circular h = 0.0						
l		Qadm (kN)	Q (kN)	u (m)	Reinforcement	Qadm / Q	
	S1	429.69	(KN) 162.85	3.8264	(m), (cm2) / n x ¢	2.64 > 1	
Ш	31	429.09	102.85	3.6204		2.04 > 1	

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	γc	Punching capacity
EN 1992-1-1:2004	1.5	430 kN
AC:2008		
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	1.5	430 kN
NA2005		
NS-EN 1992-1-	1.5	430 kN
1:2004/NA:2008		
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:	<i>ρ</i> _{<i>x</i>} = <i>ρ</i> _{<i>y</i>} =0.0077	

MATERIAL:

Concrete: C20/25

FORCES IN THE NODE:

 $\begin{array}{ll} \mbox{Vertical force:} & \mbox{N} = 192 \mbox{ kN} \\ \mbox{Moments:} & \mbox{M}_x = 24 \mbox{ kN} \\ \mbox{M}_y = 40 \mbox{ kN} \end{array}$

CALCULATION OF PUNCHING CAPACITY:

$$V_{c} = k\beta(1+50\rho)udf_{ctd} = 210kN$$

$$k = 1.6 - d[m] = 1.391$$

$$d = 0.209m$$

$$\rho = 0.0077$$

$$u = 2(c_{x} + d + c_{y} + d) = 2.036m$$

$$c_{x} = c_{y} = 0.3m$$

$$f_{ctd} = f_{ctk} / \gamma_{c} = 1.0MPa$$

$$f_{ctk} = 1.5MPa$$

$$\gamma_{c} = 1.5$$

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

$$\left(1+1.5\frac{e}{\sqrt{A_{u}}}\right)$$

$$e = \sqrt{e_{x}^{2} + e_{y}^{2}} = 0.243m$$

$$e_{x} = M_{y} / N = 0.125m$$

$$e_{y} = M_{x} / N = 0.208m$$

(2.38) (pc = 2500 kg/m³)



$A_{\mu} = 0.426m^2$

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	

न Plate ar	nd Shell R	einforcer	ment		- 0 8	
Punching]					
Verificat	ion points			Point grouping		
Name:	S1 3	30x30	\rightarrow	List:	•	
			>>	New	Delete	
Ne	w]	Delete		Maximum punching force	e (kN)	
Position	0.00					
x = 0.0	· · ·	= 0.000	0	Node number:		
Punching: from top -						
- Hea	d			Dimensions (cm	1	
Тур			67 H	a= 0.0		
0	\bigcirc rectangular $b = 0.0$					
circular h = 0.0						
	Qadm (kN)	Q (kN)	u (m)	Reinforcement (m), (cm2) / n x ø	Qadm / Q	
S1	210.61	192.00	2.0360		1.10 > 1	

Fig. 2.1. Punching calculations dialog.

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

Autodesk [and other products] are registered trademarks or trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and/or other countries. All other brand names, product names, or trademarks belong to their respective holders. Autodesk reserves the right to alter product and services offerings, and specifications and pricing at any time without notice, and is not responsible for typographical or graphical errors that may appear in this document. © 2024 Autodesk, Inc. All rights reserved.

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]
	cy=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:	$M_x = 24 \text{ kN}$
	$M_{y} = 40 \text{ kN}$

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:

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

$$u = 4.2264m$$

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

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

$$W_x = 0.5c_y^2 + c_yc_x + 4c_xd + 16d^2 + 2\pi dc_y = 1.706$$

$$W_y = 0.5c_x^2 + c_xc_y + 4c_yd + 16d^2 + 2\pi dc_x = 1.881$$

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

$$Q = v_{Ed} \cdot A_u = 342kN$$
$$A_u = ud = 0.883m^2$$

59



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	F	Robot
Punchi	ng force		342 kN	34	45 kN
📻 Plate and She	Il Reinforcen	nent			- 0 X
Punching					
Verification po	ints		Point grouping		
Name:	S1 50x30	\rightarrow	List:		•
		>>	New		Delete
New	Delete		Maximum pun	ching force	e (kN)
Туре:	unknown	-	0.00		
Position (m) x = 0.0000 Punching:	x = 0.0000 y = 0.0000 Node number:				
Head Type rectangular circular Dimensions (cm) a = 0.0 b = 0.0 h = 0.0 h = 0.0					
Qad (kN		u (m)	Reinforcen (m), (cm2) /		Qadm / Q
S1 436.	47 344.87	4.2264			1.27 > 1
				Close	Help

Fig. 3.1. Punching calculations dialog.

ANALYSIS OF RESULTS FOR NADs:

ſ

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.

> Autodesk [and other products] are registered trademarks or trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and/or other countries. All other brand names, product names, or trademarks belong to their respective holders. Autodesk reserves the right to alter product and services offerings, and specifications and pricing at any time without notice, and is not responsible for typographical or graphical errors that may appear in this document. © 2024 Autodesk, Inc. All rights reserved.

60



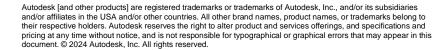
Code	Punching capacity
EN 1992-1-1:2004	345 kN
AC:2008	
PN-EN 1992-1-1:2008	345 kN
UNI-EN 1992-1-1	345 kN
EN 1992-1-1 DK NA:2007	345 kN
BS EN1992-1-1:2004	345 kN
NA2005	
NS-EN 1992-1-	345 kN
1:2004/NA:2008	
NF EN 1992-1-1/NA:2007	345 kN

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]
	$I_y = 6.00 [m]$
slab thickness:	h=24.0 [cm]
effective depth (average):	d=21.0 [cm]
column section:	40x40 [cm]

REINFORCEMENT:

reinforcement ratio:	ρ _x =ρ _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

62



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.

🛃 New Thi	ckness	
Homogene	ous Orthotropic	
	h	
:		
	h240 Color:	Auto -
Label:		
	onstant Th = 24	.0 (cm)
🔘 🖉 Va	ariable along a line	
🔘 Va	ariable on a plane	
	Point coordinates	Thicknesses
P1:	(m) 0.0000; 0.0000; 0.0000	(cm)
P2:	0.0000; 0.0000; 0.0000	
P3:	0.0000; 0.0000; 0.0000	
	eduction of the 1.00	×lg >>
	Parameters of foundation	elasticity
<u>M</u> aterial:	C20/	′25 •]
	Add Close	Help

Fig. 4.1. Slab thickness dialog



Iob Preferences				? 💌
🖙 层 🗙 关 🛛 DEFAU	ILTS			•
	• 8 •			
- Steel and timber s	Database	Database Name	Database Description	
Vehicle loads Standard loads Building soils Bolts Anchor bolts Wire fabrics Wire fabrics The sign codes The sign codes	EN 1992-1-1 EC2 - ICEL EC2 - ITALI NEN-EN PN 2002_#			4
🚔 <u>O</u> pen default par	ameters			
Save current paramet	ers as default	ОК	Cancel	Help

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

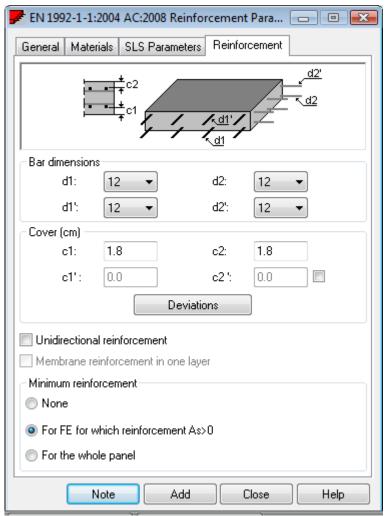
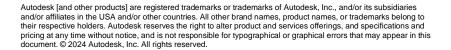


Fig. 4.3 Definition of covers of reinforcement



Reinforcemen	t Pattern - EN 1992-1	-1:2004	4 AC:2008		X
General Bars	Wire fabric reinf. Cor	istructio	onal reinf. Shapes		
-Bottom reinforce	ment		Top reinforcemen	t	ОК
Diameter			Diameter		Cancel
Direction X	Auto 👻	>>	Direction X	12 🔻	
Direction Y	Auto 🔹		Direction Y	12 🔹	Help
- Spacing (cm)-			- Spacing (cm)		•
Direction >	(10.0	<<	Direction X	6.0	Save As
Direction Y	' 10.0		Direction Y	6.0	
					Delete
Preferred reinfor	cement spacing				
Direction X—					
Maximum	40.0 cm		Minimum	3.0 cm	
-Direction Y-					
📃 Maximum	45.0 cm		Minimum	3.0 cm	

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 cm², while in Robot it is 4.14 cm² (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





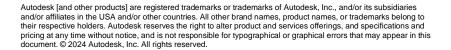
न Plate ar	nd Shell R	leinforce	ment			
Punching						
Verifical	rification points Point grouping]	
Name:	S1 4	40x40	\rightarrow	List:		-
			>>	New		Delete
Ne	w	Delete		Maximum pu	nching force	e (kN)
Туре:	unki	nown	-	0.00)	
Position	n (m)		_	L		
x = 0.1	0000 у	= 0.000	0	Node numb	er:	
Punchi	ng:	from t	op	V		
		ſ			ensions (cm a = 0.0 o = 0.0 n = 0.0	
	Qadm (kN)	Q (kN)	u (m)	Reinforce (m), (cm2)		Qadm / Q
S1	664.67	664.67	4.2389	L1=0.1050 L2 A=4.14 / 1 L1=0.2428 L2 A=4.14 / 1	2=0.1050 15 ¢6 2=0.2428	1.00 > 1
L					Close	Help

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	2 perimeters
AC:2008	$A=4.14 \text{ cm}^2$
PN-EN 1992-1-1:2008	2 perimeters
	$A=4.14 \text{ cm}^2$
UNI-EN 1992-1-1	2 perimeters
	$A=4.14 \text{ cm}^2$
EN 1992-1-1 DK NA:2007	2 perimeters
	$A=3.99 \text{ cm}^2$
BS EN1992-1-1:2004	2 perimeters
NA2005	A=4.14 cm^2
NS-EN 1992-1-	3 perimeters
1:2004/NA:2008	$A=4.14 \text{ cm}^2$
NF EN 1992-1-1/NA:2007	2 perimeters
	$A=3.72 \text{ cm}^2$

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.



67

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.:

a Preferences			? ×
E X X STANDARI Languages General Parameters View Parameters Desktop Settings Toolbar & Menu Printout Parameters Advanced	D <u>R</u> egional settings: <u>W</u> orking language: <u>P</u> rintout language:	Eurocode	
☑ Update Preferences on exit		Accept Cancel	Help

B. Job Preferences

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

Job Preferences			? <mark>X</mark>
	RO		•
⊡ ··· Units and Formats ···· Dimensions ···· Forces	Steel/Aluminum structures:	EN 1993-1:2005	
Other Unit Edition	Steel connections:	EN 1993-1-8:2005	·
Materials ⊕ Databases	Timber structures:	PN-8-03150	•
Design codes	<u>R</u> C structures:	PN-B-03264 (2002)	•
	<u>G</u> eotechnical:	PN-81/B-03020	•
		More codes	
🙀 <u>O</u> pen defaul	t parameters		
Save current para	ameters as default	OK Cancel H	elp

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 <u>code</u>, 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:*



70

Configuration of Code List					X
Codes:				Current codes:	
Steel / aluminum		•		Set as current	
Code	Country	*		Code	
AL76	France	=		EN 1993-1:2005	
ANSI/AISC 360-05	USA	_	Σ	PN-90/B-03200	
ASD:1989 Ed.9th	USA				
Add80	France				
BS - EN 1993-1-1:2005	UK EC3				
BS 5950:2000	UK		S		
BS5950	UK				
BSK99	Sweden				
CAN/CSA-S16-01 + Supp. No.1 (2005)	Canada	-			
<	- .	•		< III	•
OK Cancel					<u>H</u> elp

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

Reg Configuration of Code List				×		
Codes:				Current codes:		
Steel / aluminum 🔹			•	Set as current		
Steel / aluminum Steel connections RC Timber Geotechnical Load combinations Snow/wind loads Seismic loads			2	Code BS-EN 1993-1:2005/NA:2008/AC: DS/EN 1993-1:2005/DK NA:2007/ EN 1993-1:2006/AC:2009 ENV 1993-1:1992 NAD Belgium		
ASD:1989 Ed.9th Add80 BS 5950:2000 BS-EN 1993-1:2005/NA:2008/AC:2009 BS5950	USA France UK UK EC3 UK	-	2	ENV 1993-1:1992 NAD Germany ENV 1993-1:1992 NAD Netherland NF EN 1993-1:2005/NA:2007/AC: NS-EN 1993-1:2005/NA:2008/AC: PN-EN 1993-1:2006/AC:2009		
OK Cancel				<u>H</u> elp		

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

Configuration of Code List					X
Codes:				Current codes:	
Timber		•		Set as current	
Code	Country			Code	
CB71	France			EN 1995-1:2004/A1:2008	
CB71+KERTO	France		Σ	PN-EN 1995-1:2005/A1:2008	
EN 1995-1:2004/A1:2008	Eurocode 5				
ENV 1995-1-1:1992	Eurocode				
ENV 1995-1:1992 NAD Finland	Finland EC5				
NF EN 1995-1:2005/NA:2007/A1:2008	France EC5		1		
PN-B-03150	Poland				
PN-EN 1995-1:2005/A1:2008	Poland EC5				
•		•		< III	+
OK Cancel				<u>H</u> e	lp

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

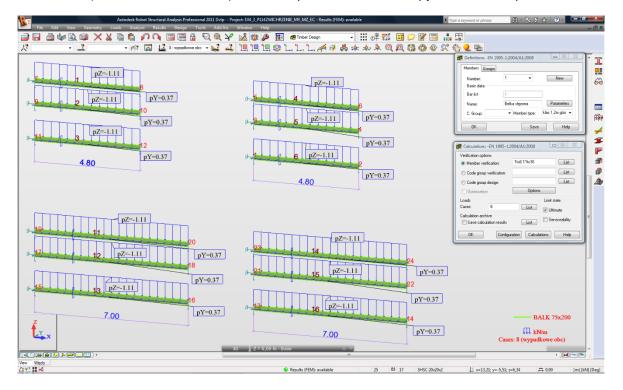
Example based on "Practical design of timber structures to Eurocode 5" Hans Larsen and Vahik Enjily File: EX_4_3p114_bending_My_Mz.rtd

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 n° 1, 4, 11, 14 are restrained at 1,2m against torsion. For load case n° 8 loads are assumed as a short-term load and are acting on the bottom (for el. n° 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-defi	ned type of	member (e a '	(timber beam)	mav	be initially	opened	
	the pre-den	neu type or	member (e.y.	unber beam j	тпау	be initially	y openeu.	

Definitions -EN 1	995-1:2004/A1:2008	
Members Groups		
N <u>u</u> mber:	11 👻	New
Basic data		
Bar list:	11	
<u>N</u> ame:	klasyczna stęzona	Parameters
C. <u>G</u> roup:	✓ Member <u>t</u> ype:	klas 1,2m gór
		Timber Member
ОК	<u>S</u> ave	Timber Column Timber Beam
		Pret drewniany

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

Member Definition - Parameter	ters - EN 1995-1:2004/A1:2008	×
Member type: Timber Beam		Save
Buckling (Y axis) Member length ly: Real	Buckling (Zaxis) Member length Iz:	Close
© Coefficient	 Coefficient 1,00 	<u>S</u> ervice
Buckling length coefficient Y:	Buckling lengt <u>h</u> coefficient Z:	<u>M</u> ore
1,00	1,00	<u>0</u> ther
Lateral buckling parameters		<u> </u>
Lat. buckling type:	Lateral buckling length coeff.	
Load level:	Id = Io Id = Io	
	eated as solid	
Method of critical stress determination - 6.3.3 : O Classic - formula [6.31]		
Eor rectangular sections - formula [6.32]		
S <u>e</u> rvice class:	1 •	Help

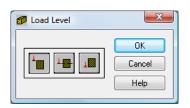
Type a new name in *Member type* editable field. Next, change the parameters to meet the initial data requirements of the structure. Set the following lateral-buckling parameters:

switch on the appropriate Lateral buckling type icon;

💋 Lateral B	luckling Type	×
₹11 ×	Beam with pinned supports Beam with fixed supports Cantilever Without lateral buckling	OK Cancel Help



select the appropriate Load level icon



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

Member Definition - Ad	dditional Param	neters X
Load parameters Load type:		OK Cancel
Section parameters Anet/Agross ratio	1,00	Help
Additional conditions for ro		

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

💋 Load T	уре	×
	Moment at the end Uniform load Concentrated force	OK Cancel Help

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

 \rightarrow 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:

💋 Latera	Lateral Buckling Length Coefficient		
	ld = 2lo		
	ld = lo He	lp)	
	ld = 0.5 lo		
	<u>l</u> d = 1,00 lo		
X			
I	Intermediate bracings		



74

The second one opens BUCKING TYPE DIAGRAMS dialog:

Buckling Type Diagrams	×
$\begin{array}{c c} \hline \\ \hline \\ 10 \\ \hline \\ 0.5 \\ \hline \\ \hline \\ 0.7 \\ \hline \\ 0.7 \\ \hline \\ 0.9 \\ \hline \\ \hline \\ 0.9 \\ \hline \\ \hline \\ 0.9 \\ \hline \\ $	OK Cancel
<u>C</u> omplex member Batten type: packs	Help
Connection: glued	
Batten coordinates: peal peal peal peal	

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

🗲 Internal bracings			x	
Internal bracings				
Coordinates of the existing bracings	buckling-	plower flange Automatic detection of bracings		
Define manually coordinates of the existing bracings 1,20; 2,40; 3,60] m	Add bracings at points where adjoining elements occur Add bracings at points where bending moments equal zero Bracing detection preview For member no.:		
Buckling coefficients of component segments]	For load case: <u>1 c własny</u>		
OK Cancel		Help		

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.



lem <u>b</u> er type: klas 1,2m góra	st	Save		
uckling (Yaxis) Member length ly: Beal	Buckling (Zaxis) Member length Iz: Real	Close		
Coefficient	 Coefficient 	<u>S</u> ervice		
Buckling length coefficient Y:	Buckling length coefficient Z:	<u>M</u> ore		
1,00 L	1,00	<u>O</u> ther		
ateral buckling parameters		<u>F</u> ire		
	Lateral buckling length coeff. Upper flange			
oad level:	d = (ld1,ld2,) ld = lo			
Double sections tr	eated as solid			
ethod of critical stress determina <u>C</u> lassic - formula [6.31] <u>For rectangular sections - form</u>		Note		
S <u>e</u> rvice class:	1	Help		
	ſ	Definitions -EN	1995-1:2004/A1:2008	x
		Members Groups	8	
<u>per</u> of the member m ned to the appropria		Number:	1 - N <u>e</u> w	
ember type.		- Basic data		
		<u>B</u> ar list:	1	
s very important whe	en	<u>N</u> ame:	Belka stęzona Parameters	

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

In the CALCULATIONS dialog set the following:

- -> Verification options list of verified members,
- -> Loads cases list of chosen loads
- -> Limit state
- ->Configuration.

Calculations -EN 1995-1:	2004/A1:2008	
Verification options		
Member verification:	1to6 11to16	List
🔘 Code group verification:		List
🔘 Code group <u>d</u> esign:		List
	Options	
Loads		Limit state
Cases: 8	List	🔽 <u>U</u> ltimate
Calculation archive	List	Serviceability
OK	Iration Calcula	itions Help



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

Configuration	<u> </u>	Ru Load Case Classification - Duration
Calculation points	ОК	Case list:
Number of points: 3	Cancel	Number Name A D Class Number Name OK
Characteristic points Options		4 wiatr >>
Calculation parameters		5 s nierzutowane → 8 wypadkowe obc
Efficiency ratio: 1,00	Help	
Maximum slenderness: 210,00		
Components of complex bars are not taken into account		Load class according to duration: 4. Short-term 1. Permanent
Calculations: fire impact		2. Long-term 3. Medium-term
Load case classification - duration		4. Short-term 5. Instantaneous
Exclude internal forces from calculations		
Units of results Code Robot		
Camber		
Take the deflections from the following case into consideration:		
1 c własny		

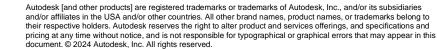
 - in LOAD CASE CLASSIFICATION-DURATION dialog, assign "Load class according to duration" from selection list to the number of case list; for this particular example 4th "short-term" load case was selected and LOAD CASE CLASSIFICATION-DURATION dialog after the introduced changes looks as follows:

Rue Load Case Classification - D	uration				×
Case list:					
Number Name		Class	Number	Name 🔶	ОК
		4	4	wiatr	Cancel
		4	5	s nierzub 😑	Cancer
		4	8	wypadkc 📮	
•	• <<	•	II	Þ	Help
Load class according to duration:					
1. Permanent	•				

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.

esults Messages								Calc. Note Close
Member	Т	Section	Material	Lay	Laz	Ratio	Case	Help
1 Belka stęzona	ОК	BALK 75x200	C16	83.14	221.71	0.89	8 wypadkowe obc	
2 Belka obc. górą	ОК	BALK 75x200	C16	83.14	221.71	0.89	8 wypadkowe obc	Ratio
3 Belka wolnopodp	ОК	BALK 75x200	C16	83.14	221.71	0.89	8 wypadkowe obc	Analysis Map
4 Belka stęzona	СК	BALK 75x200	C16	83.14	221.71	0.89	8 wypadkowe obc	Calculation points
5 Belka obc. górą	OK	BALK 75x200	C16	83.14	46.19	0.89	8 wypadkowe obc	Division: n = 3
6 Belka wolnopodp	OK.	BALK 75x200	C16	83.14	221.71	0.89	8 wypadkowe obc	Extremes: none
11 klasyczna stęzona	8	BALK 75x200	C16	121.24	323.32	1.89	8 wypadkowe obc	Additional: none
12 klas obc. górą	8	BALK 75x200	C16	121.24	323.32	1.89	8 wypadkowe obc	
13 klasycz obcdół	8	BALK 75x200	C16	121.24	323.32	1.89	8 wypadkowe obc	
14 uproszcz stężona	8	BALK 75x200	C16	121.24	323.32	1.89	8 wypadkowe obc	
15 Belka obc. górą	8	BALK 75x200	C16	121.24	46.19	1.89	8 wypadkowe obc	
16 uproszcz obcdól	8	BALK 75x200	C16	121.24	323.32	1.89	8 wypadkowe obc	



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

RESULTS - Code - EN 1995-1:2004/A1:2008		
BALK 75x200 ▼ Bar: 1 Belk	a stęzona s: 2 / x = 0.50 L = 2 40 m 8 wypadkowe obc	ОК
Simplified results Detailed results		<u>C</u> hange
CALCULATION STRESSES	ALLOWABLE STRESSES	
Sig_m,y,d = 3.19/500.00 = 6.38 MPa Sig_m,z,d = 1.07/187.49 = 5.68 MPa	f m.y.d = 11.08 MPa f m.z.d = 12.72 MPa	Eorces Det <u>a</u> iled
FACTORS AND ADDITIONAL PARAMETERS km = 0.70 kh = 1.15 kmod =	- 0.90 Ksys = 1.00	Calc. Note
LATERAL BUCKLING lef = 1.48 m Sig_cr = 70.43	Lambda_rel m = 0.48 3 MPa k crit = 1.00	Help
BUCKLING Y	BUCKLING Z	нер
	\mathbf{X}	
RESULTS Sig_m.y.d/f m.y.d + km*Sig_m.z.d/f m.z.d = 6.38/ Sig_m.y.d/(kcrit*f m.y.d) = 6.38/(1.00*11.08) = 0.5		

Detailed results tab

ALK 75x200	<u>A</u> uto	Bar: Point / Coo Load case:	1 Belka stęzona rdinate: 2 / x = 0.50 L = 2.40 m	ction OK	° • •	OK
implified results	Detailed results	:				<u>C</u> hang
Symbol	Value	Unit	Symbol description	Section	*	
Sig_m,z,d	5.68	MPa	Left edge normal stress due to Mz	[6.1.6]		
fm,y,d	11.08	MPa	Allowable normal stress from bending	[6.1.6]		
fm,z,d	12.72	MPa	Allowable normal stress from bending	[6.1.6]		<u>F</u> orce
		Fac	tors and additional parameters			Det <u>a</u> il
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)]		Calc. N
kmod	0.90		Modification factor depending on time of load action	[3.1.3]		L <u>a</u> ic. N
km	0.70		Interaction factor due to bending [6.1.6.(2)]			
Ksys	1.00		System coefficient	[6.7]		
		Paran	neters of lateral buckling analysis			Help
Method of cri	tical stress de	eterminatio	n - 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)]	E	
k crit	1.00		Lateral buckling factor	[6.3.3.(4)]		
			Ratio:			
Delta	0.89		Ratio between normal and allowable stresses	Section OK		

Autodesk [and other products] are registered trademarks or trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and/or other countries. All other brand names, product names, or trademarks belong to their respective holders. Autodesk reserves the right to alter product and services offerings, and specifications and pricing at any time without notice, and is not responsible for typographical or graphical errors that may appear in this document. © 2024 Autodesk, Inc. All rights reserved.



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: <i>EN 1995-1:20</i> ANALYSIS TYPE: Me			
CODE GROUP: MEMBER: 1 Belka stę		COORDINA	TE: $x = 0.50 L = 2.40 m$
LOADS: Governing Load Case: 8	3 wypadkowe obc		
MATERIAL C16 gM = 1.30 f v,k = 1.80 MPa E 0,05 = 5400.00 MPa	f m,0,k = 16.00 MPa f t,90,k = 0.50 MPa G moyen = 500.00 MPa	f t,0,k = 10.00 MPa f c,90,k = 2.20 MPa Service class: 1	f c,0,k = 17.00 MPa E 0,moyen = 8000.00 MPa Beta c = 1.00
ht=20.0 cm	AMETERS: BALK 75x20	0	
bf=7.5 cm tw=3.8 cm tf=3.8 cm	Ay=40.91 cm2 Iy=5000.00 cm4 Wely=500.00 cm3	Az=109.09 cm2 Iz=703.10 cm4 Welz=187.49 cm3	Ax=150.00 cm2 Ix=2148.0 cm4
STRESSES Sig_m,y,d = MY/Wy= 3. Sig_m,z,d = MZ/Wz= 1.0		ALLOWABLI f m,y,d = 11.08 f m,z,d = 12.72	
Factors and additiona $km = 0.70$ $kh = 1.1$		Ksys = 1.00	
LATERAL BUC lef = 1.48 m Sig_cr = 70.43 MPa	KLING PARAMETERS: Lambda_rel m = 0.48 k crit = 1.00		
BUCKLING PARAMET	ERS:	About Z axis:	
	ULAS: Sig_m,z,d/f m,z,d = 6.38/11.0 = 6.38/(1.00*11.08) = 0.58 -		.89 < 1.00 (6.11)
Section OV III			

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.

RESULTS - Code - EN 19	95-1:2004/A1:2008	
BALK 50x100	Bar: 1 Belka stęzona Point / Coordinate: 2 / x = 0.50 L = 2.40 m Load case: 8 wypadkowe obc	ОК
BALK 50x115	suits	Change
BALK 50x125 BALK 50x140 BALK 50x150 BALK 50x160 BALK 50x200 BALK 50x200 BALK 50x255 BALK 50x250 BALK 50x250 BALK 63x150 BALK 63x150 BALK 63x160 BALK 63x175	S ALLOWABLE STRESSES 8.29 MPa fm,y,d = 12.01 MPa 5.57 MPa fm,z,d = 13.80 MPa	Eorces Det <u>a</u> iled
BALK 63x200	VAL PARAMETERS	
BALK 63x225 BALK 63x250 BALK 75x150 BALK 75x160 BALK 75x160 BALK 75x175	1.25 kmod = 0.90 Ksys = 1.00	Calc. Note
BALK 75x200 BALK 75x200 BALK 75x225 BALK 75x250 BALK 100x200	lef = 1.28 m Lambda_rel m = 0.48 Sig_cr = 68.56 MPa k crit = 1.00	Help
BALK 100x225 BALK 100x255 BALK 100x250 RESULTS		
Sig_m,y,d/fm,y,d + km*	Sig_m,z,d/f m,z,d = 38.29/12.01 + 0.70°25.57/13.80 = 4.48 > 1.00 (6.11) = 38.29/(1.00°12.01) = 3.19 > 1.00 (6.33)	

Auto Bar: 1 Bell BALK 53x225 ▼ Load case:	section DK e: 2 / x = 0.50 L = 2.40 m 8 wypadkowe obc
Simplified results Detailed results	
CALCULATION STRESSES	ALLOWABLE STRESSES
Sig. m,y,d = 3.19/531.56 = 6.00 MPa	fm,y,d = 11.08 MPa
Sig_m.z,d = 1.07/148.83 = 7.16 MPa	fm.z.d = 13.18 MPa
FACTORS AND ADDITIONAL PARAMETERS km = 0.70 kh = 1.19 kmod	= 0.90 Ksys = 1.00
LATERAL BUCKLING	
lef = 1.52 m	Lambda_rel m = 0.60
Sig_cr = 44.3	7 MPa k crit = 1.00
BUCKLING Y	BUCKLING Z
\mathbf{X}	\mathbf{X}
RESULTS km*Sig_m,y,d/fm,y,d + Sig_m,z,d/fm,z,d = 0.70° Sig_m,y,d/(kcrit*fm,y,d) = 6.00/(1.00*11.08) = 0.	

The results for the newly selected section are presented below.





TIMBER	STRUCTURE CALCUL	ATIONS for BALK	63x225
CODE: EN 1995-1:200 ANALYSIS TYPE: Mer			
CODE GROUP: MEMBER: 1 Belka stęz	rona POINT: 2		
LOADS: Governing Load Case: 8			
MATERIAL C16 gM = 1.30 f v,k = 1.80 MPa E 0,05 = 5400.00 MPa			Beta c = 1.00
, −, , ,	RAMETERS: BALK 63x225		
bf=6.3 cm	Ay=31.02 cm2	Az=110.78 cm2	Ax=141.80 cm2
tw=3.1 cm tf=3.1 cm	Iy=5980.10 cm4 Wely=531.56 cm3	Iz=468.80 cm4 Welz=148.83 cm3	Ix=1544.5 cm4
STRESSES Sig_m,y,d = MY/Wy= 3.1 Sig_m,z,d = MZ/Wz= 1.07		ALLOWA f m,y,d = f m,z,d =	ABLE STRESSES = 11.08 MPa = 13.18 MPa
Factors and additional $km = 0.70$ k		0.90 Ksys = 1.00	
┎┷╪┷┰	EKLING PARAMETERS: Lambda_rel m = 0. k crit = 1.00	.60	
BUCKLING PARAMET	ERS:		bout Z axis:
Sig_m,y,d/(kcrit*f m,y,d)	JLAS: ig_m,z,d/f m,z,d = 0.70*6.00/11. = 6.00/(1.00*11.08) = 0.54 < 1.00	$\begin{array}{rcl} 08+7.16/13.18=& 0.92 &< 1.00\\ 0 & (6.33) \end{array}$. ,

Section OK !!!

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

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

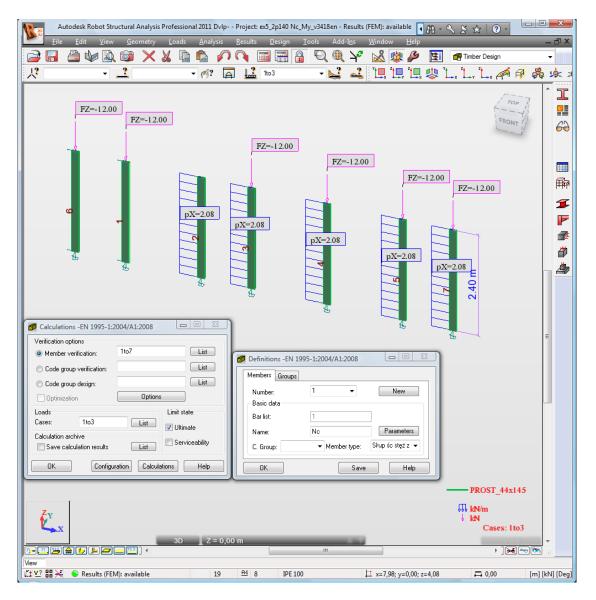
Example based on "Practical design of timber structures to Eurocode 5" Hans Larsen and Vahik Enjily File: EX_5_2p140_Nc_My.rtd

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 [and other products] are registered trademarks or trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and/or other countries. All other brand names, product names, or trademarks belong to their respective holders. Autodesk reserves the right to alter product and services offerings, and specifications and pricing at any time without notice, and is not responsible for typographical or graphical errors that may appear in this document. © 2024 Autodesk, Inc. All rights reserved.



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.

Definitions -EN	1995-1:2004/A1:2008	
Members Groups		
N <u>u</u> mber: Basic data	1 🔹	New
<u>B</u> ar list:	1	
<u>N</u> ame:	Nc	Parameters
C. <u>G</u> roup:	✓ Member type:	Słup śc stęż z 👻
ОК	Save	Timber Member Timber Column Timber Beam Słup drewniany

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

💋 Member Definition - Paramet	ters - EN 1995-1:2004/A1:2008	×				
Member type: Timber Column		Save				
Buckling (Yaxis) Member length ly:	Buckling (Z axis) Member length Iz:	Close				
 <u>R</u>eal 1,00 Coefficient 	○ Real Coefficient ^{1,00}	<u>S</u> ervice				
Buckling length coefficient Y:	Buckling length coefficient Z:	<u>M</u> ore				
1,00	1,00	<u>O</u> ther				
Lateral buckling parameters	<u> </u>					
Lat. buckling type:						
Load level:	ld = lo ld = lo					
Double sections to	Double sections treated as solid					
Method of critical stress determina	Note					
Olassic - formula [6.31]						
─ Eor soft timber - formula [6.32]						
S <u>e</u> rvice class:	1 •	Help				

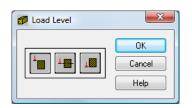
Type a new name in the *Member type* editable field. Next, change the parameters to meet the initial data requirements of a structure, e.g.:

switch on the appropriate Lateral buckling type icon;

💋 Lateral E	🗊 Lateral Buckling Type		
X I I F	Beam with pinned supports Beam with fixed supports Cantilever Without lateral buckling	OK Cancel Help	



select appropriate Load level icon



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

Load parameters OK Load type: Cancel Section parameters Anet/Agross ratio Anet/Agross ratio 1.00 Help Additional conditions for round sections Unidirectional bending	Member Definition - A	Additional Param	ieters X
Anet/Agross ratio 1.00 Help Additional conditions for round sections	·		
	·	1,00	Help

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

💋 Load Type		×
	ient at the end orm load centrated force	OK Cancel Help

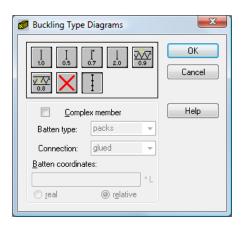
• define bracings for Lateral buckling and Buckling.

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,

💋 Latera	Buckling Length Coef	ficient 💌
	ld = 2lo	OK Cancel
	ld = lo	Help
	ld = 0.5 lo	
	<u>I</u> d = 1,00 lo	
X		
I	Intermediate bracings	

the second opens > BUCKING TYPE DIAGRAMS dialog.



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

Internal bracings			
Test for member: 3 Nc+My bez wyb z-z		ng-lower flange	
Buckling Y Buckling Z Lateral buckling-upper flange Latera	-	Automatic detection of bracing	21
Define manually coordinates of the existing bracings			ere adjoining elements occur
U Denne manually coordinates or the existing bracings	_		
	m	Add bracings at points whe	ere bending moments equal
I ieal I ieal		Bracing detection preview	
Basic scheme of a member		For member no.:	3 Nc+My bez wyb z-z
		For load case:	1 STA1
Buckling coefficients of component segments			m
			11
OK Cancel			Help

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



85

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:

💋 Member Definition - Paramet	ters - EN 1995-1:2004/A1:2008	×
Member type: Słup śc stęż z-	z	Save
Buckling (Y axis) Member length ly: © <u>R</u> eal © Cgefficient	Buckling (Z axis) Member length Iz: Real Coefficient	Close Service
Buckling length coefficient Y:	Buckling lengt <u>h</u> coefficient Z:	<u>M</u> ore
	Lateral buckling length coeff. <u>Upper flange</u> Id = Io Id = Io	<u>F</u> ire
Method of critical stress determina © Classic - formula [6.31] © For soft timber - formula [6.32]	ation - 6.3.3 :	Note
S <u>e</u> rvice class:	2 🗸	Help

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

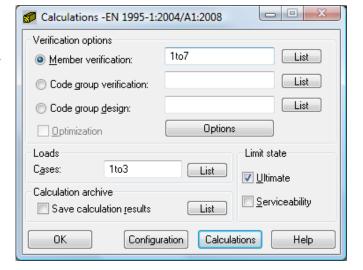
→ it is very important when verifying different member types

Definitions -EN 19	995-1:2004/A1:2008	
Members Groups		
N <u>u</u> mber:	3 🗸	New
Basic data		
<u>B</u> ar list:	3	
<u>N</u> ame:	Nc+My bez wyb z-z	Parameters
C. <u>G</u> roup:	✓ Member type:	Słup śc stęż z 👻
ОК	<u>S</u> ave	Help

In CALCULATIONS dialog	, set the following:
------------------------	----------------------

- -> Verification options list of verified members
- -> Loads cases list of chosen loads
- -> Limit state
- -> Configuration.

AUTODESK





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

Configuration	<u> </u>	Rig Load Case Classification - Duration	X
Configuration Calculation points Number of points: Calculation points: Calculation parameters Efficiency ratio: Calculation parameters: Efficiency ratio: Calculations: fire impact Calculations: fire impact Load case classification - duration Exclude internal forces from calculations: Units of results Cade Protot Robot	Cancel Help	Case list: Number Name > Class Number Name	OK Cancel Help
Code Probot Camber Take the deflections from the following case into consideration: 1 STA1			

 - 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:

Run Load Case Classification - Durati	on				×
Case list:					
Number Name	\supset	Class	Number	Name	ОК
	>>	1 5	1 2	STA1 wiatr	Cancel
4 III +	<<	ا	I		• Help
Load class according to duration:					
1. Permanent	•				

Start verification by pressing Calculations button in CALCULATIONS dialog.

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

	N 1995-1:2004/A1:20	80	- Member Verifi	cation (UL	S) 1to7				
Re	sults Messages								Calc. Note Close
	Member		Section	Material	Lay	Laz	Ratio	Case	Help
	1 Nc	0K	PROST_44x145	C16	57.34	188.95	0.36	1 STA1	
	2 My	0K	PROST_44x145	C16	57.34	188.95	0.79	2 wiatr	Ratio
3	Nc+My bez wyb z-z	0K	PROST_44x145	C16	57.34	188.95	0.91	3 KOMB1	Analysis Map
	4 bez zwich	<mark>0K</mark>	PROST_44x145	C16	57.34	94.48	0.93	3 KOMB1	Calculation points
	5 Nc+My wyb + zw	8	PROST_44x145	C16	57.34	94.48	1.05	3 KOMB1	Division: n = 3
	6 Nc	8	PROST_44x145	C16	57.34	188.95	2.89	1 STA1	Extremes: none
	7 Nc+My wyb + zw	Ж	PROST_44x145	C16	57.34	47.24	0.91	3 KOMB1	Additional: none

Autodesk [and other products] are registered trademarks or trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and/or other countries. All other brand names, product names, or trademarks belong to their respective holders. Autodesk reserves the right to alter product and services offerings, and specifications and pricing at any time without notice, and is not responsible for typographical or graphical errors that may appear in this document. © 2024 Autodesk, Inc. All rights reserved.



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

RESULTS - Code - EN 1995-1:20	004/A1:2008		
	Bar: 3 Nc+Mybezwy Point / Coordinate: 2 / Load case: 3 ł		ОК
Simplified results Detailed results			<u>C</u> hange
CALCULATION STRESSES Sig_c.0.d = 12.00/63.80 = 1.88 M Sig_m.y.d = 1.50/154.18 = 9.73 M		ALLOWABLE STRESSES fc.0.d = 14.38 MPa fm.y.d = 13.63 MPa	Eorces Det <u>a</u> iled
FACTORS AND ADDITIONAL P/ km = 0.70 kh = 1.28 LATERAL BUCKLING	kmod = 1.10	Ksys = 1.00	Calc. Note Parameters
	lef = 2.45 m Sig. cr = 20.79 MPa	Lambda_relm = 0.88 k.crit = 0.90	
BUCKLING Y LY = 2.40 m LFY = 2.40 m Lambda Y = 57.34 RESULTS Sig_c.0,d/{kc,y*f c,0,d} + Sig_m Sig_m.y,d/{kcn*f m.y,d} = 9.73/	Lambda_rel Y = 1.02 ky = 1.10 kcy = 0.67 uy.d/f m.y.d = 1.88/(0.67*14	BUCKLING Z	Help

Detailed results tab

	<u>A</u> uto	Bar: 3 Point / Coo Load case:	Nc+My bez wyb z-z	Section OK	• • •	
mplified results	Detailed results	3				<u>C</u> har
Symbol	Value	Unit	Symbol description	Section	^	
km	0.70		Interaction factor due to bending	[6.1.6.(2)]		
Ksys	1.00		System coefficient	[6.7]		
			Buckling parameters			<u>F</u> orc
About the Y a	xis of cross-s	ection			1	Deta
LY	2.40	m	Member length	[6.3.2]		
LFY	2.40	m	Buckling length	[6.3.2]		
Lambda Y	57.34		Member slenderness	[6.3.2]		
Sig c,crit,y	16.21	MPa	Critical stress (buckling)	[6.3.2.(1)]		Calc.
Lambda_rel	1.02		Relative slenderness (buckling)	[6.3.2.(1)]		
ky	1.10		Slenderness factor	[6.3.2.(3)]		Param
kcy	0.67		Reduction factor due to compression	[6.3.2.(3)]		
		Param	neters of lateral buckling analysis			He
Method of cri	itical stress d	etermination	1 - Classic - formula (6.31)			
lef	2.45	m	Lateral buckling length	[6.3.3]		
Sig_cr	20.79	MPa	Critical stress (lateral buckling)	[6.3.3]		
Lambda_rel	0.88		Relative slenderness (lateral buckling)	[6.3.3.(2)]		
k crit	0.90		Lateral buckling factor	[6.3.3.(4)]	E	
			Ratio:			
Delta	0.91		Ratio between normal and allowable stresses	Section OK		



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 1995-1:20 ANALYSIS TYPE: Me					
-	bez wyb z-z		COINATE: $x = 0.50 L = 1.20 m$		
LOADS: Governing Load Case: 3					
$\begin{array}{l} \textbf{MATERIAL} C16 \\ gM = 1.30 \\ f \ v,k = 1.80 \ MPa \\ E \ 0,05 = 5400.00 \ MPa \end{array}$	f t,90,k = 0.50 MPa	f c,90,k = 2.20 M			
SECTION PAR ht=14.5 cm	AMETERS: PROST_4	4x145			
bf=4.4 cm tw=2.2 cm tf=2.2 cm	Ay=14.85 cm2 Iy=1117.83 cm4 Wely=154.18 cm3	Az=48.95 cm2 Iz=102.93 cm4 Welz=46.79 cm3	Ax=63.80 cm2 Ix=333.0 cm4		
STRESSES Sig_c,0,d = N/Ax = 12.00/63.80 = 1.88 MPa Sig_m,y,d = MY/Wy= 1.50/154.18 = 9.73 MPa		ALLOWABLE STRESSES f c,0,d = 14.38 MPa f m,y,d = 13.63 MPa			
Factors and additiona $km = 0.70$ $kh = 1.2$		•			
	KLING PARAMETERS Lambda_rel m = 0.88 k crit = 0.90				
BUCKLING PARAMET i_{10} About Y axis: LY = 2.40 m Lambda_rel Y = 1.02 LFY = 2.40 m	FERS: Lambda Y = 57.34 ky = 1.10 kcy = 0.67	About Z a	kis:		
VERIFICATION FORM Sig_c,0,d/(kc,y*f c,0,d) + Sig_m,y,d/(kcrit*f m,y,d)	Sig_m,y,d/f m,y,d = 1.88	· · · · · · · · · · · · · · · · · · ·	3.63 = 0.91 < 1.00 (6.23)		

Section OK !!!



COMPARISON:

e.g. for member n ° 3 \rightarrow for the axial load Nc and My moment

verifications parameters, interaction expres	Robot	Handbook	
λy - member slenderness ky - slenderness factor kcy - reduction factor due to compression k _{mod} f c,o,d - design compression strength f m,y,d - design bending strength due to My σ c,o,d - design compression stress σ m,y,d - design bending stress due to My	[MPa] [MPa] [MPa] [MPa]	57,34 1,097 0,671 1,1 14,38 13,63 1,88 9,73	57,3 1,097 0,671 1,1 14,38 13,54 1,88 9,73
ratio from (6.23) $\rightarrow \sigma$ 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.

