Autodesk Robot Structural Analysis VERIFICATION MANUAL

For EU Codes

March 2012

© 2012 Autodesk, Inc. All Rights Reserved. Except as otherwise permitted by Autodesk, Inc., this publication, or parts thereof, may not be reproduced in any form, by any method, for any purpose. Certain materials included in this publication are reprinted with the permission of the copyright holder.

Disclaimer

THIS PUBLICATION AND THE INFORMATION CONTAINED HEREIN IS MADE AVAILABLE BY AUTODESK, INC. "AS IS." AUTODESK, INC. DISCLAIMS ALL WARRANTIES, EITHER EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO ANY IMPLIED WARRANTIES OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE REGARDING THESE MATERIALS.

Trademarks

The following are registered trademarks of Autodesk, Inc., in the USA and/or other countries: Autodesk Robot Structural Analysis, Autodesk Concrete Building Structures, Spreadsheet Calculator, ATC, AutoCAD, Autodesk, Autodesk Inventor, Autodesk (logo), Buzzsaw, Design Web Format, DWF, ViewCube, SteeringWheels, and Autodesk Revit. All other brand names, product names or trademarks belong to their respective holders.

Third Party Software Program Credits

ACIS Copyright© 1989-2001 Spatial Corp. Portions Copyright© 2002 Autodesk, Inc. Copyright© 1997 Microsoft Corporation. All rights reserved. International CorrectSpell™ Spelling Correction System© 1995 by Lernout & Hauspie Speech Products, N.V. All rights reserved. InstallShield™ 3.0. Copyright© 1997 InstallShield Software Corporation. All rights reserved. PANTONE® and other Pantone, Inc. trademarks are the property of Pantone, Inc.© Pantone, Inc., 2002. Portions Copyright© 1991-1996 Arthur D. Applegate. All rights reserved. Portions relating to JPEG © Copyright 1991-1998 Thomas G. Lane. All rights reserved. Portions of this software are based on the work of the Independent JPEG Group. Portions relating to TIFF © Copyright 1997-1998 Sam Leffler. © Copyright 1991-1997 Silicon

Graphics, Inc. All rights reserved.

Government Use

Use, duplication, or disclosure by the U.S. Government is subject to restrictions as set forth in FAR 12.212 (Commercial Computer Software-Restricted Rights) and DFAR 227.7202 (Rights in Technical Data and Computer Software), as applicable.

INTRODUCTION	1
STEEL	2
1. EUROCODE 3 (EN 1993-1-1:2005)	3
VERIFICATION EXAMPLE 1 - AXIAL COMPRESSION VERIFICATION EXAMPLE 2 - AXIAL COMPRESSION WITH BUCKLING VERIFICATION EXAMPLE 3 - COMBINED COMPRESSION AND BENDING	4 7 10
VERIFICATION EXAMPLE 4 - BENDING WITH LATERAL BUCKLING VERIFICATION EXAMPLE 5 - COMBINED BI-AXIAL BENDING AND COMPRESSION	13 17
CONCRETE	21
1. EUROCODE 2 EN 1992-1-1:2004 AC:2008 - RC BEAMS	22
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	23 27 31 33 38 40
2. EUROCODE 2 EN 1992-1-1:2004 AC:2008 - RC COLUMNS	41
VERIFICATION EXAMPLE 1 - COLUMN SUBJECTED TO AXIAL LOAD AND UNI-AXIAL BENDING	42 48
3. EUROCODE 2 EN 1992-1-1:2004 AC:2008 - RC SLABS (PUNCHING)	.49
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	50 55 57 60 65
TIMBER	66
1. EUROCODE 5: DESIGN OF TIMBER STRUCTURES; EN 1995-1:2004/A1:2008	67
GENERAL REMARKS VERIFICATION PROBLEM 1 BENDING ABOUT TWO MAIN AXES WITH LATERAL BUCKLING VERIFICATION PROBLEM 2 COMBINED COMPRESSION AND BENDING ABOUT ONE MAIN AXIS	68 70 80

INTRODUCTION

This verification manual contains numerical examples for structures prepared and originally calculated by **Autodesk Robot Structural Analysis version 2013**. 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 (signed further as "Robot").

Each example contains the following parts:

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

STEEL

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

VERIFICATION EXAMPLE 1 - Axial compression

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

TITLE:

Axial compression (Example 6.2 page 44).

SPECIFICATION:

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



SOLUTION:

Define a new type of member. For analysed 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 twelvth 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	Member type: Column1	Save
Number: 1 New Basic data	Buckling (Y axis) Buckling (Z axis) Member length ly: Member length lz: Beal 1.00 Coefficient 1.00 Buckling length coeff. Y: Buckling length coeff. Y: 1.00 Non-sway Buckling curve Y auto V	Close
	Elexural-torsional buckling of monosymmetric sections	
Buckling Diagrams	Lateral buckling parameters Lateral buckling length coefficient Load level: Upper flange Lower flange Lor = lo Lor = lo	More
$ \begin{array}{c c} \hline \\ 1 \\ 1 \\ 2 \\ 0.8 \end{array} \end{array} \begin{array}{c} 1 \\ 1 \\ 1 \\ 1 \\ 1 \\ 1 \\ 1 \\ 1 \\ 1 \\ 1 $	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 lateral restraints [6.3.2.4] Kfl = [1.10]	
C Sway structure	Additional sets of member parameters	
<u>N</u> on-sway structure	Complex sections: Complex sections: Complex sections:	
	Ihin-walled sections: Thin-walled	Note
	Eire analysis parameters:	Help

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

🗾 Calculations - EN 1993-1:2005
Verification options
<u>M</u> ember verification: <u>1</u> <u>List</u>
Code group verification:
Code group design: List
Optimization Options
Loads Limit state
Cases: 1
Calculation archive
Save calculation results List
OK Configuration 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 analysed member.

1	EN 1993-1:2005 - Member Verification (ULS) 1									
	Results Message:	C <u>a</u> lc. Note	Close							
	Member	Section	Material	Lay	Laz	Ratio	Case			Help
	1 Bar 1	UC 254x254x7	S355	36.13	61.74	1.00	1 STA1		- D - K	
									Analysis	Map
									Calculation p Division:	n = 3
									Extremes:	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 - FN 1993-1:2005		🖉 RESULTS - Co	de - EN 1993-1:	2005			
Auto Bar 1 Bar 1 / x = 0.00 L = 0.00 m UC 254-254-73 Point / Coordinate: 1 / x = 0.00 L = 0.00 m	ОК	UC 254x254x7	Auto	Bar: Point / Co Load case	1 Bar 1 Section 0K scriniste: 1 / x = 0.00 L = 0.00 m x 1 STA1		ОК
Simplified results Detailed results	Diverge	Simplified result:	Detailed results	1			Change
FORCES N.E.d.+ 3305 kN	12	Symbol	Values	Unit	Symbol description Section	-	
Nc.Rd = 3305 M Nb.Rd = 3305 M				Cross	-section properties: UC 254x254x73		
10,10 - 300,101	Eorces	Ax	93.100	cm2	Cross-section area	- 11	Eorces
	Dutid	Ay	75.883	cm2	Shear area - Y-axis		Durbel
Class of section = 2	necheo.	Az	25.792	cm2	Shear area - Z-axis		Decigieo
LATERAL BUCKLING		bx .	57.600	cm4	Torsional constant		
XLT = 1.00		ly .	11410.000	cm4	Moment of inertia of a section about the Y-axis		
	7e	12	3908.000	cm4	Moment of inertia of a section about the Z-axis		
BUCKLING Y BUCKLING Z	Cale Mote 1	Wply	990.000	cm3	Plastic section modulus about the Y (major) axis		Cale Mate
	Cgc. Note	Wpiz	463.000	cm3	Plastic section modulus about the Z (minor) axis		Cgrc. rvore
	Parameters	h	25.40	cm	Height of cross-section		Parameters
		b	25.40	cm	Width of cross-section		
	Refe	tf	1.42	cm	Flange thickness		Hub 1
SECTION CHECK	nep	tw	0.86	cm	Web thickness		nep
NEd/Nc.Bd = 1.00 (1.00 624 1)		N N	11.07	cm	Radius of gyration - Y-axis		
		12	6.48	cm	Radius of gyration - Z-axis		
MEMBER STABILITY CHECK					Material:		
Not enalized		Name			\$355 (\$355)	-	
		fy	355.00	MPa	Design yield strength of material (3.2)	1	

STEEL DESIGN

DE GROUP: MEMBER: 1 Bar 1 0.00 m	POINT: 1	CC	DORDINATE: x = 0.00 L =
L OADS: Governing Load Case	: 1 STA1		
MATERIAL: \$355 (\$355) fy=	= 355.00 MPa		
	PARAMETERS: UC 254x254	4x73	
n=25.40 cm	gM0=1.00	gM1=1.00 A $z=25.702 \text{ cm}^2$	$\Delta x = 03.100 \text{ cm}^2$
W=0.86 cm	$I_{y}=11410\ 000\ cm4$	$I_{z=3908,000 \text{ cm}4}$	$Ix=57\ 600\ cm4$
f=1.42 cm	Wply=990.000 cm3	Wplz=463.000 cm3	
NTERNAL FORCE N,Ed = 3305 kN Nc,Rd = 3305 kN Nb,Rd = 3305 kN	S AND CAPACITIES:		
			Class of section = 2
LATERAL B	UCKLING PARAMETERS:		
	METERS:		
About Y axis	S:	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 analysed 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 N <u>u</u> mber: 1 N <u>e</u> w	Member type: Column2 Buckling (Y axis) Buckling (Z axis) Member length ly: Member length lz:	Save Close
Basic data Bar list: Name: Bar 1 C Group: Column1 Column1	C Beal C Real C Coefficient 1.00 Buckling length coeff. Y: Buckling length coeff. Z: 1.00 1.00 I.00 1.00	
OK Save Help	Buckling curve Y auto Buckling curve Y auto Buckling curve Z auto	
Buckling Diagrams \square 1 1 1 1 1 2 1 1 <tr< th=""><th>Lateral buckling Lateral buckling Load level: Lateral buckling Load level: Lateral buckling Lor = lo Lor = lo C General method [6.3.2.2] Lambda LT.0 = O Detailed method [6.3.2.3] Beta = O.75</th><th><u>M</u>ore</th></tr<>	Lateral buckling Lateral buckling Load level: Lateral buckling Load level: Lateral buckling Lor = lo Lor = lo C General method [6.3.2.2] Lambda LT.0 = O Detailed method [6.3.2.3] Beta = O.75	<u>M</u> ore
C Sway <u>s</u> tructure Non-sway structure	Additional sets of member parameters Interain restraints [B,3,2,4] Additional sets of member parameters Interain terms and displacements: Complex sections: Complex Thin welled sections: Thin welled	
	Fire analysis parameters: Fire	Help

In the CALCULATIONS dialog set *Member Verification* option for member 1 and switch off *Limit State* – *Serviceability* (only Ultimate Limit state will be analysed). 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
Cases: 1	List Ultimate
Calculation archive	Serviceshilitu
Save calculation results	
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 analysed member.

1	EN 1993-1:2005 - Member Verification (ULS) 1									
	Results Message:	s							C <u>a</u> lc. Note	(<u>C</u> lose
	Member	Section	Material	Lay	Laz	Ratio	Case			Help
	1 Bar 1	CHS 244.5x10	S275	48.21	48.21	0.89	1 STA1		- Dalia	
									Analysis	Map
									Calculation po Division:	n = 3
									Extremes:	none
	1								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 - Code - EN 1993-1:2005	
Auto Bar: 1 Bar 1 Section 0K CHS 244 5x10 Edd case: 1 / x = 0.00 L = 0.00 m	ОК	Auto Bar 1 Bar 1 Point / Coodinate. 1 / x = 0.00 L = 0.00 m Load Coat. 1 \$ TA1	Section OK
Simplified results Detailed results	Distric	implified results	Diange
FORCES N E4 - 163.0 kN N c Ru = 2026 8 kN		Symbol Values Unit Symbol description	Section A
Nb.Rd = 1836.5 kN	Former	Av 73.700 cm2 Cross-section area	Forces
		Av 46.919 cm2 Shear area - Y-axia	
Class of section = 1		Az 66.919 cm2 Shear area - Z-axis	
LATERAL BUCKLING		N 10150.000 cm4 Torsional constant	
XLT = 1.00		Iv 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-axia	
BUCKING 7	(margine and the second se	Woly 550.238 cm3 Plastic section modulus about the Y (major) axis	
t lv=400m Lan v=0.56 L lz=400m Lan z=0.56	Calc. Note	Wplz 550.236 cm3 Plastic section modulus about the Z (minor) axis	Calc. Note
te Lory=400m Xy=0.91 te Lorg=400m Xg=0.91	Parameters	h 24.45 cm Height of cross-section	Parameters
Lang - 40,21		b 24.45 cm Width of cross-section	
		tf 1.00 cm Flange thickness	
	Help	tw 1.00 cm Web thickness	Help
SECTION CHECK N C44No R4 - 0.90 / 1.00 JE 2.4 (1)		ry 8.30 cm Radius of ovration - Y-axis	
HARMONE WELLIN DESIT		rz 8.30 cm Radius of gyration - Z-axis	
MEMBER STABILITY CHECK		Material:	
NEd/Nb.Rd + 0.89 < 1.00 (6.3.1.1.01)		Name \$275 (\$275)	
		fy 275.00 MPa Design yield strength of material	(3.2)

STEEL DESIGN

CODE: EN 1993-1:2 ANALYSIS TYPE: M	005, Eurocode 3: De	sign of steel structures.	
CODE GROUP: MEMBER: 1 Bar 1 0.00 m	POINT:	1 CO	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 24 gM0=1.00 Ay=46.919 cm2 Iy=5073.000 cm4 Wply=550.236 cm3	44.5x10 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 PARAMETE	RS:	
BUCKLING PARAME \downarrow_{10} About Y a Ly = 4.00 m Lcr,y = 4.00 m Lamy = 48.21	TERS: xis: Lam_y = 0.56 Xy = 0.91	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 < Lam N,Ed/Nb,Rd = 0.89 < 1.0	MULAS: 00 (6.2.4.(1)) f member: nbda,max = 210.00 00 (6.3.1.1.(1))	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 analysed 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 twelvth 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 Number: 1 Basic data Bar list: Name: Bar 1 C. Group: Member type:	Member Definition - Parameters - EN 1993-1:2005 Member Lype: Column1 Buckling (Y axis) Save Member length ly: Beal © Coefficient 1.00 Buckling length coeff. Y: I.00 I.00 I.00 Non-sway Buckling curve Y Buckling curve Y Non-sway Buckling curve Z Total	×]
OK Save Help	Image: section]
C Sway <u>structure</u>	○ 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 lateral restraints [6.3.2.4] kfl = 1.10 Additional sets of member parameters Service ☑ Limit deflections and displacements: Service	
• Non-sway structure	Complex sections:	
	Inin-walled sections: Thin-walled	
	Eire analysis parameters: 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 analysed). 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						
Cases: 1	List 🔽 Ultimate						
Calculation archive							
Save calculation results	List						
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 analysed member.

1	EN 1993-1:2005 - Member Verification (ULS) 1										
	Results Messages	s							C <u>a</u> lc. Note	<u>C</u> lose	
	Member	Section	Material	Lay	Laz	Ratio	Case			Help	
	1 Bar 1	UB 457x191x9	S235	20.91	92.31	1.00	1 STA1		D V		
									Analysis	Map	
									Calculation po Division:	n = 3	
									Extremes:	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		ESULTS - Cod	e - EN 1993-1:2	005			
Auto Back 1 Back Section DK UB 457x191x98 Point / Coordinate 1 / x = 0.00 L = 0.00 m	0K	UB 457x191x98	Auto	Bar: Point / Co Load case	1 Bar 1 Section 0K scalar 1 / x = 0.00 L = 0.00 m x 1 STA1	•	ОК
Simplified results Detailed results	Distrige	Simplified results	Detailed results				Dhange
FORCES NEd=1400.0 kN MyEd=-342.2 kN/m	2	Symbol	Values	Unit	Symbol description Section	-	
NC.Hd = 2337.5 kN My.p.Hd = 523.0 kN m Nb.Rd = 2937.5 kN My.c.Rd = 525.0 kN m				Cross	-section properties: UB 457x191x58		
My.N.Fid = 342.5 kN*m	Eorces	Ax	125.000	cm2	Cross-section area		Eorces
	Datala	Ay	78.511	cm2	Shear area - Y-axis		Dataland
Class of section = 1	neilinea	Az	55.655	cm2	Shear area - Z-axis		Leideo
LATERAL BUCKLING	2	x	121.000	cm4	Torsional constant		
XLT = 1.00		ly .	45730.000	cm4	Moment of inertia of a section about the Y-axis		
	2	lz .	2347.000	cm4	Moment of inertia of a section about the Z-axis		
BUCKLING Y BUCKLING Z	Calo Moto I	Wply	2234.000	cm3	Plastic section modulus about the Y (major) axis		Cala Moto
	Cgc.rece	Wplz	379.000	cm3	Plastic section modulus about the Z (minor) axis		Cgec. Note
	Parameters	h	46.74	cm	Height of cross-section		Parameters
		b	19.28	cm	Width of cross-section		
	Halo I	tf	1.96	cm	Flange thickness		Halo
SECTION CHECK	nep	tw	1.14	cm	Web thickness		nep
MyEdMyNRd = 100 < 1.00 6.2.9.1.[2]		ry .	19.13	cm	Radius of gyration - Y-axis		
		12	4.33	cm	Radius of gyration - Z-axis		
MEMBER STABILITY CHECK					Material:		
Not enalyzed		Name			\$235 (\$235)		
	2	fy	235.00	MPa	Design yield strength of material (3.2)	1	
						_	1

STEEL DESIGN

CODE GROUP: MEMBER: 1 Bar 1 0.00 m	POINT: 1	C	OORDINATE: x = 0.00 L =
LOADS: Governing Load Case:	1 STA1		
MATERIAL: S235 (S235) fy = 2	35.00 MPa		
	RAMETERS: UB 457x191x	98	
h=46.74 cm	gM0=1.00	gM1=1.00	A
b=19.28 cm	Ay=/8.511 cm2 Jy=45730.000 cm4	$AZ=33.633 \text{ cm}^2$ $Iz=23.47,000 \text{ cm}^4$	Ax=125.000 cm2 Ix=121.000 cm4
tf=1.96 cm	Wply=2234.000 cm3	Wplz=379.000 cm3	1x-121.000 cm4
INTERNAL FORCES	AND CAPACITIES:		
N,Ed = 1400.0 kN	My,Ed = -342.2 kN*m		
Nc,Rd = 2937.5 kN	My,pl,Rd = 525.0 kN*m		
Nb,Rd = 2937.5 kN	My,c,Rd = 525.0 kN*m		
	My,N,Rd = 342.5 kN*m		Class of section $= 1$
	CKLING PARAMETERS:		
BUCKLING PARAME	TERS:		
About Y axis:		About Z axis:	
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	MULAS: 00 (6.2.4.(1)) < 1.00 (6.2.5.(1)) < 1.00 (6.2.9.1 (2))		

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 analysed 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.

🗲 Definitions - EN 1993-1:2005	Member Definition - Parameters - EN 1993-1:2005
Members Groups	Membertype: Beam1 Save
Number: 1 New Basic data	Buckling (Y axis) Member length ly: Beal Coefficient 1.00 Buckling length coeff. Y: 1.00 Non-sway Buckling curve Y Buckling length coeff. Z: 1.00 Non-sway Buckling curve Z Buckling
Lateral Buckling Length Coefficient Lcr = 2 lo Lcr = lo Lcr = lo Help Lcr = 0.5 lo Lcr = 1.00 lo	Lateral buckling parameters ✓ Lateral buckling Lateral buckling Lateral buckling Logad level: Upper flange Lor = (Lor1, Lor2,) Lor = lo © 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 = lateral restraints [6.3.2.4] Additional sets of member parameters ✓ Limit deflections and displacements: Service © Complex sections:
Intermediate bracings	Inin-walled sections: Thin-walled Note Ere analysis parameters: Ere Help

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.



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.

Angles in tension [8: 3.5) Conjucted by one bolt row Number of bolts m Distances between bolts p1: 3.00 cm Distances between bolts and angle gdge e2 Flipes Vield strength: Carego f_ys = 235.00 MPs Additional conditions for round pipes	vuriform or varying linearly moment uniform load - simply supported beam uniform load - fixed beam concentrated force in the center fixed beam concentrated force in the center fixed beam concentrated force in the center fixed beam concentrated forces in the center fixed beam	OK Cancel 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 analysed). Call configuration dialog and set number of calculation points to 101.

🗲 Calculations - EN 1993-1:2005	Configuration
Verification options	Calculation points
<u>Member verification:</u> <u>1</u> List	Number of points:
C Code group verification:	Characteristic points
O Code group design:	Calculation parameters
Optimization	Efficiency ratio: 1,00 Help
Loads Limit state	Maximum slenderness: 210,00
Calculation archive	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 second

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 analysed member.

1	EN 1993-1:2005	i - Member Verifi			_ 🗆 🗡					
	Results Message:	C <u>a</u> lc. Note	(<u>C</u> lose							
	Member	Section	Material	Lay	Laz	Ratio	Case			Help
I	1 Bar 1	UB 762x267x1	S275	35.35	193.55	0.91	1 STA1		- D - K	
									Analysis	Map
									Calculation p Division:	points n = 101
									Extremes:	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 - Cod	de - EN 1993-1:	2005			_ 🗆 🗙
Auto Bar. 1 Bar 1 Vib 752-057x173 Point / Coordinate. 54 / x = 0.53 L = 5.72 m Load case: 1 STA1	ОК	UB 762x267x17	Auto 3 💌	Bar: Point / Co Load case	1 Bar 1 Section DK codinate: 54 / x = 0.53 L = 5.72 m x 1 5TA1		ОК
Simplified results Detailed results	Dimpe	Simplified results	Detailed results	1			Change
FORCES MyEd = 1226.7 MVm Morel Rd = 1200.5 MVm		Symbol	Values	Unit	Symbol description Section	-	
My.c.Rd = 1703.6 kN*m Vz.Ed = -251.5 kN	1			Cross	-section properties: UB 762x267x173	_	I
Vz,c,Rd = 1825.9 kN	Eorces	Ax	220.000	cm2	Cross-section area	-	Eorces
Mb.Hd = 1401.2 kN/m	Detailed	Ay	121.931	cm2	Shear area - Y-axis		Detailed
CABLE OF RECOON = 1		Az	115.002	cm2	Shear area - Z-axis	-	
LATENAL BUCKLING LATENAL BUCKLING Mrc= \$2119 kM/m Dowel Tub VI T=0.92		×	267.000	cm4	Torsional constant	-100	
- In In II - 052 611 - 077		N	205300.000	cm4	Moment of inertia of a section about the Y-axis	-100	
		12	6850.000	cma	Moment of inertia of a section about the 2-axis	-100	
BUCKUNG Z	Calc. Note	Wpty	6195.000	cm3	Plastic section modulus about the Y (major) axis	-100	Calc. Note
×	Decementaria	mpiz	70.000	ema	Plastic Section modulus about the 2 (minor) axis	-111	Buumuhuu
	r diditotest		76.20	cm .	With of cross-section	-	r di di licitei s
	12 X		2 16	cm	Flance thickness	-	I
	Help	Day .	1.43	cm	Web Dickness	-100	Help
SECTION CHECK		N	30.55	cm	Radius of ovration - Y-axis	-	
VEEAVec Rd = 0.14 / 1.00 (6.6.01)		12	5.58	cm	Radius of gyration - Z-axis		
MEMBER STABILITY CHECK		<u> </u>			Material:		
MuEdAbbBd=0.91<1.00 (6.3.2.1.01)		Name			\$275 (\$275)		
		fy	275.00	MPa	Design yield strength of material (3.2)	-	
	J					_	

STEEL DESIGN

CODE: EN 1993-1:20 ANALYSIS TYPE: Mer	05, Eurocode 3: Design comber Verification	of steel structures.	
CODE GROUP: MEMBER: 1 Bar 1 5.72 m	POINT: 54		COORDINATE: x = 0.53 L =
LOADS: Governing Load Case: 1	STA1		
MATERIAL: S275 (S275) fy = 27	5.00 MPa		
SECTION PAR h=76.20 cm b=26.67 cm tw=1.43 cm tf=2.16 cm	AMETERS: UB 762x267x gM0=1.00 Ay=121.931 cm2 Iy=205300.000 cm4 Wply=6195.000 cm3	173 gM1=1.00 Az=115.002 cm2 Iz=6850.000 cm4 Wplz=807.000 cm3	Ax=220.000 cm2 Ix=267.000 cm4
INTERNAL FORCES A	ND CAPACITIES:		
	My,Ed = 1276.7 kN*m My,pl,Rd = 1703.6 kN*m My,c,Rd = 1703.6 kN*m Mb,Rd = 1401.2 kN*m		Vz,Ed = -251.5 kN Vz,c,Rd = 1825.9 kN Class of section = 1
LATERAL E z = 1.00 Lcr,upp=5.10 m	BUCKLING PARAMETER Mcr = 4311.9 kN*m Lam_LT = 0.63	S: Curve,LT - b fi,LT = 0.77	XLT = 0.82
BUCKLING PARAMET	ERS:		
About Y axis:		About Z axi	S:
VERIFICATION FORM Section strength check: My,Ed/My,c,Rd = 0.75 < Vz,Ed/Vz,c,Rd = 0.14 < 1 Global stability check of n My,Ed/Mb,Rd = 0.91 < 1.	JLAS: 1.00 (6.2.5.(1)) .00 (6.2.6.(1)) <i>nember:</i> 00 (6.3.2.1.(1))		
Section OK !!!			

COMPARISON: Critical segment CD

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

VERIFICATION EXAMPLE 5 - Combined bi-axial bending and compression

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

TITLE:

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

SPECIFICATION:

The model represents a column in a multistory building. The column frame is moment resisting in-plna 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 analysed 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*.

Members Groups Number: 1 Basic data Basic data Bar list: 1 Name: Bar 1 Bar 1 Parameters C. Group: Member type:	🗾 Definitions - EN 1993-1:2005	🗲 Buckling Diagrams	×
OK Sway structure	Definitions - EN 1993-1:2005 Members Members Groups Number: 1 Basic data Barlist: 1 Name: Bar1 C. Group: Member type: Column1 OK	Buckling Diagrams $ \begin{array}{c c} \hline \\ 10 \\ 10 \\ 10 \\ 10 \\ 10 \\ 10 \\ 10 \\ 10$	Cancel Help

Set Lateral buckling checkbox.

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

ቻ Member Definition - Parame	ters - EN 19	93-1:2005	×
Member <u>type:</u> Column1			Save
Buckling (Yaxis) Member length lv:	Buckling (Z Mem	axis) ber length lz:	Close
C Beal © Coefficient	⊂ Re <u>a</u> l ⊙ C <u>o</u> efficia	ent 1,00	
Buckling length coeff. Y:	Buckling	g length coeff. Z:	
Non-sway Buckling curve Y auto	Bucking <u>c</u> u	Non-sway Irve Z auto 💌	
Elexural-torsional buckling of m	nonosymmetric	c sections	
Lateral buckling parameters	Lateral buckli	ing length coefficient	More
Load level:	Upper flange Lcr = lo	Lower flange	
• <u>G</u> eneral method [6.3.2.2]	Lambd	la LT,0 = 0,40	
Detailed method [6.3.2.3]	<u>B</u> eta =	0,75	
Simplified method for beams w lateral restraints [<u>6</u> .3.2.4]	^{iith} k <u>f</u> l =	1,10	
Additional sets of member paramet	ers		
Limit deflections and displacer	nents:	<u>S</u> ervice	
Comple <u>x</u> sections:		<u>C</u> omplex	
Thin-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 analysed). Now, start the calculations by pressing *Calculations* button.

🗲 Calculations - EN 1993-1	:2005 📃 🖂 🔀
Verification options	1 List
<u>Member verification:</u>	
C Code group verification:	
	Options
Cases: 1	List Vitimate
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 analysed member.

5 EN 1993-1:200	EN 1993-1:2005 - Member Verification (ULS) 1											
Results Message	es							C <u>a</u> lc. Note	Close			
Member	Section	Material	Lay	Laz	Ratio	Case			Help			
1 Bar 1	UC 305x305x2	S 275	20.30	51.55	0.97	1 STA1		- Patia				
								Calculation p	Map			
								Extremes: 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 1993-14	2005				🖉 RESULTS - Co	de - EN 1993-1:	2005			
₩40 UC 305×305×240	Bac 1 Bar 1 Point / Coordinate: 3 / Load case: 11	/ x = 1.00 L = 4.20 m STA1	Section OK	<u>ОК</u>	UC 305×305×2	Auto 40 💌	Bar: Point / Co Load case	1 Bar 1 Section 0K codinate 3 / x = 1.00 L = 4.20 m x 1 STA1		0K
Simplified results Detailed results	1			Diage	Simplified result	Detailed results	1			Change
FORCES N.Ed = 3440 kN N.E.R.d = 8415 kN N.b.R.d = 6640 kN	My,Ed = -420 kN°m My,pl,Rd = 1167 kN°m My,pl,Rd = 1167 kN°m	M2.Ed = 110 kN/m M2.pl.Rd = 535 kN/m M2.c.Rd = 535 kN/m	Vy.Ed = -26 kN Vy.c.Rd = 3957 kN Vz.Ed = -200 kN		Symbol	Values	Unit	Symbol description Section -section properties: UC 305x305x240	-	
	My,N,Rd = 774 kN'm Mb,Rd = 1150 kN'm	Mz,N,Rid = 503 kN*m	Vz.c.Rd = 1372 kN Class of section = 1	Eorces Detgled	Ax Ay	306.000 249.236	cm2 cm2	Cross-section area Shear area - Y-axis Shear area - Z-axis	1	Eorces Detgiled
LATERAL BUCKLING z = 0.00 Loc.low=4.20 =	Mcr = 16778 kN*m = Lom_LT = 0.26	Curve_LT - a 6LT = 0.54	XLT = 0.99		k k ly	1271.000 64200.000 20310.000	cm4 cm4	Torsional constant Moment of inertia of a section about the Y-axis		
BUCKUNG Y Ly = 4.20 m Lor.y = 2.94 m	Lam_y = 0.23 Xy = 0.99	BUCKLING Z Lz = 4.20 m Lor.z = 4.20 m	Lan_z = 0.59 ⊠z = 0.79	Cglc. Note Paraméters	Wply Wplz h	4243.000 1945.000 35.26	cm3 cm3 cm	Plastic section modulus about the Y (major) axis Plastic section modulus about the Z (minor) axis Height of cross-section		Calc. Note Parameters
Lany - 20.30	kzy = 0.79	Lane - 51.55	kzz = 0.79	Help	b tf tw	31.79 3.77 2.30	om om	Width of cross-section Flange thickness Web thickness		Help
MyEd/My.c.Rd + MzEd/Mz.c. VzEd/Vz.c.Rd = 0.15 < 1.00	Ad = 0.57 < 1.00 (6.2.5 (1)) (6.2.6 (1))				rz	14.48 8.15	cm cm	Radius of gyration - Y-axis Radius of gyration - Z-axis		
MEMBER STABILITY CHECK	00 Lanz = 51.55 c Lan	max + 210.00 STABLE						Material:		
N,Ed/D(z'N,Rk/gM1) + kzy/My	(Ed/pdLT*My(Rk/gM1) + kzz*	Mz,Ed/(Mz,Rk/gM1) = 0.97 < 1.0	10 (6.3.3.(4))		Name fy	275.00	MPa	S 275 (S275) Design yield strength of material (3.2)	-	

STEEL DESIGN

CODE: <u>EN 1993-1</u> : ANALYSIS TYPE: 1	2005, Eurocode 3: Design of Member Verification	of steel structures.	
CODE GROUP: MEMBER: 1 Bar 1	POINT: 3	COORD	EXAMPLE: $x = 1.00 L = 4.20 m$
LOADS: Governing Load Case:	1 STA1		
MATERIAL: S 275 (S275) fy =	- 275.00 MPa		
	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	Iy=64200.000 cm4	Iz=20310.000 cm4	Ix=1271.000 cm4
tf=3.77 cm	Wply=4243.000 cm3	Wplz=1945.000 cm3	
INTERNAL FORCES	AND CAPACITIES:		
N,Ed = 3440 kN	My, Ed = -420 kN*m	Mz, Ed = 110 kN*m	Vy,Ed = -26 kN
Nc,Rd = 8415 kN	My,pl,Rd = 1167 kN*m	Mz,pl,Rd = 535 kN*m	Vy,c,Rd = 3957 kN
Nb,Rd = 6640 kN	My,c,Rd = 1167 kN*m	Mz,c,Rd = 535 kN*m	Vz,Ed = -200 kN
	My,N,Rd = 774 kN*m Mb,Rd = 1150 kN*m	Mz,N,Rd = 503 kN*m	Vz,c,Rd = 1372 kN
			Class of section $= 1$



Section OK !!!

COMPARISON:

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

CONCRETE

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

VERIFICATION EXAMPLE 1 - Dimensioning reinforcement in rectangular section at bending

Example based on:

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

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

DESCRIPTION OF THE EXAMPLE:

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

GEOMETRY:

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

MATERIAL:

Concrete:	C25/30	
α_{cc} = 0.85		
Steel:	fvk=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*).







Ī		Case	Distributed	Nature	Li	Positi	Coord.	Load directi	Load factor	Coordinates	Projection	x1 (m)	p1
		number	load		st	on	syste			coordinates			(kN/m)
	1	DL1	uniform	dead loa	1	upp	Local	Z	1.35	absolute	Not projected		23.70
l	*												

Fig. 1.2 Loads and the calculation model

Job Preferences				? 💌
DEFAL	ULTS Database EN 1992-1-1 EC2 - ICEL EC2 - ITALI NEN-EN PN 2002_#	Database Name	Database Description	
👻 <u>O</u> pen default pa	rameters			
Save current paramet	ters as default		Cancel	Help

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

Job Preferences		ŀ	? 💌
🚅 🖶 🗙 关 📑	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	
🔍 🛛 Open defa	ult parameters	<u></u> 010 00000	
Save current pa	arameters as default	OK Cancel He	elp

🎦 Partial Factors for a Code EN 1992-1-1:2004 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)		
DN EN 1992 1 1/2009	11	k4 (redistribution)	Auto	1992-1-1 5.5 (4)	=	
FINEN 1332-1-1.2008	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	acc	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						
			ОК	Cancel He	lp	

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 reinf. 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	$\gamma_{\rm s}$	α_{cc}	bottom reinf. A_{sl} -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 are based on the last edition with corrections EN1992-1-1:2004 AC:2008. However, the last correction does not introduce any changes within the range of the calculations presented in the examples.

DESCRIPTION OF THE EXAMPLE:

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

RESULTS OF LONGITUDINAL REINFORCEMENT (REINFORCEMENT FOR BENDING) CALCULATION:

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

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

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





ANALYSIS OF RESULTS FOR NADs:

The presented example has been calculated for the parameters assumed in [1]. As discussed above, although the example is calculated for general edition of the code [2], the authors of [1] use the partial factor α_{cc} = 0.85. The default value for the general edition of the code is α_{cc} = 1.0.

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

Code	γc	$\gamma_{\rm s}$	α _{cc}	bottom reinf. A_{sl} -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 cm^2
AC:2008					
PN-EN 1992-1-	1.4	1.15	1.0	33.09 cm^2	0.0 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 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 are based on the last edition with the corrections EN1992-1-1:2004 AC:2008. However, the last correction does not introduce any changes within the range of the calculations presented in the examples.

DESCRIPTION OF THE EXAMPLE:

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

RESULTS OF LONGITUDINAL REINFORCEMENT (REINFORCEMENT FOR BENDING) CALCULATION:

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

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

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





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	γ_{c}	$\gamma_{\rm s}$	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 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_{sI}	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 fesuits	Robot results
Handbook example	1.5	1.15	0.85	3.64 cm^2	4.34 cm^2
(general Eurocode 2					
edition with modified					
$\alpha_{cc)}$					
EN 1992-1-1:2004	1.5	1.15	1.0	4.75 cm^2	0.0 cm^2
AC:2008					
PN-EN 1992-1-1:2008	1.4	1.15	1.0	3.24 cm^2	0.0 cm^2
UNI-EN 1992-1-1	1.5	1.15	0.85	3.64 cm^2	4.34 cm^2
SFS-EN 1992-1-1	1.5	1.15	0.85	3.64 cm^2	4.34 cm^2
EN 1992-1-1 DK	1.45	1.2	1.0	4.13 cm^2	0.0 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 showthe 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: $q_L = 20 \text{ [kN/m]}$

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.



A 0.40												<u> </u>	
🖽 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:

$$V_{x=0} = q_{tot} \cdot \frac{l}{2} = 239.7kN$$
$$l = 6.8m$$

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

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

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

• CALCULATION OF SHEAR CAPACITY OF A BEAM WITHOUT SHEAR REINFORCEMENT:

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

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

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

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]:

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

z = 0.9d = 0.49m

And assuming $V_{Rd,s} = V_{x=0.4}$, the required spacing of stirrups near the support is: $s = \frac{A_{sw}}{V_{x=0.4}} z f_{ywd} \cot \theta = 0.101m$ $A_{sw} = 0.000101m^2$ (2 bars ϕ 8) $V_{x=0.4} = 211.5kN$

$$d = 0.6 - 0.059 = 0.541m$$

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

$$f_{ywk} = 500MPa$$

$$\gamma_s = 1.15$$

$$\cot \theta = 1.0$$

(set in Calcualtion options/General)

(for bottom bars at the support)

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

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

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)

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	$\gamma_{\rm s}$	α_{cc}	Shear capacity $V_{Rd,c}$	Shear capacity
				r v 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 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 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:	I=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*.





A 7.50											Δ
Case number	Distributed load	Nature	Li st	Positi on	Coord. syste	Load directi	Load factor	Coordinates	Projection	x1 (m)	p1 (kN/m)
DL1	uniform	dead loa	1	upp	Local	Z	1.35	absolute	Not projected		22.76

Fig. 5.2 Loads and the calculation model

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



Fig. 5.2 Reinforcement (5\u00f620) assumed in [1]

RESULTS OF DEFLECTION CALCULATION:

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

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

$$\xi = 0.9686$$

$$f_{I} = 2.720cm$$

$$f_{II} = 3.791cm$$

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

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

ANALYSIS OF RESULTS FOR NADs:

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

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

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

LITERATURE

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

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

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

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

Example based on:

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

* - NOTE: the reference handbook [1] is based on the edition of Eurocode 2 EN1992-1-1:2004. The calculations in program Autodesk Robot Structural Analysis 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 analyse 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 coefficient: $\phi = 2.3$ 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 toal 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)*.

묘 Job Preferences				? 💌
🖙 🖶 🗙 🔆 🛛 DEFAU	ILTS			•
•• Units and Formats ••• Materials ••• Databases	• 8 •			
	Database	Database Name	Database Description	
Vehicle loads Standard loads Building soils Bolts Anchor bolts Reinforcing bars Wire fabrics	EN 1992-1-1 EC2 - ICEL EC2 - ITALI NEN-EN PN 2002_#			4
🚔 <u>O</u> pen default par	rameters			
🖳 <u>S</u> ave current paramet	ers as default	ОК	Cancel	Help

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

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*).

$\overline{}$	De	siq	n for :	simple	bei	ndin	ġ.			
	Θ	M	y dire	ction				О М	z directi	ion
	~ ~	•		~	•					

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)*.

Job Preferences		? <mark>×</mark>		
🖙 🖶 🗙 🔆 🛛 🛤	FAULTS	•		
 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 ▼ More codes ■		
Cpen default parameters				
Save current par	ameters as default	OK Cancel Help		
Partial Factors for a Code FN 10	992-1-1:2004 AC:2008	?		

EN 1992-1-1-2004		Coefficient	Value	Code reference	•
L		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)	
DN EN 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)	Ŧ
			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
α_{m}	(-)	0.791	0.791
e _a	(cm)	2.0	2.0
K _r	(-)	1.0	1.0
K_{φ}	(-)	1.0	1.0
$1/r_0$	(1/m)	0.00863	0.00853*
$1/r_0$	(1/m)	0.00863	0.00853
с	(-)	10	10
e_2	(cm)	13.7 (14.1)**	14.0
\overline{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) nor (3) can be applied). It naturally leads to the greater (in this particular case by 20%), but at the same time safer, value of total moment.

ANALYSIS OF RESULTS FOR NADs:

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

Code	γ _c	α_{cc}	Design moment M_{Ed}
EN 1992-1-1:2004	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 stiffenes *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 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	[cm²/m]
reinforcement ratio:	ρ _x =ρ _y =0.0077	

MATERIAL:

Concrete:	fck = 15	[MPa]
		L 1

IMPORTANT STEPS:

In the Structure model/Geometry view, define the slab with the supporting column in the middle. The slab shoud 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 databse, 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 strucutre 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.

🛃 New Thic	kness 🗖 🗖 🗙
Homogene	ous Orthotropic
ł	
1	
Label:	h240 Color: Auto 🗸
💿 Co	nstant Th = 24.0 (cm)
🔘 Vai	riable along a line
🔘 Va	riable on a plane
	Point coordinates Thicknesses (m) (cm)
P1 :	7.0000; 2.0000; 0.0000
P2:	7.0000; 2.0000; 0.0000
P3:	7.0000; 2.0000; 0.0000 0.0
Re	duction of the 1.00 ×1g >>
	Parameters of foundation elasticity
<u>M</u> aterial:	C20/25 👻
	Add Close Help

Fig. 1.1 Slab thicknes

Reg Material Definition	2 ×					
<u>Name:</u>	Description: EC2 Concrete C12/15					
Elasticity Young modulus, <u>E</u> : Poisson ratio, <u>v</u> : Shear modulus, <u>G</u> :	27000.00 (MPa) 0.2 Characteristic 11250.00 (MPa) Sample: Cylindrical					
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

📝 EN 1992-1-1:2004 AC:2008 Reinforcement Par 🗖 🔲 🗮 🏹				
General Mate	rials SLS Parar	meters Reinfo	prcement	_
	• + c1 • + c1	्रिया रिया	<u>d2'</u> <u>d2</u>	
-Bar dimension	18			5
d1:	12 🔹	d2:	12 🔻	
d1':	12 🗸	d2':	12 🔻	
Cover (cm) -				51
c1:	1.9	c2:	1.9	
c1':	0.0	c2 ':	0.0	
)eviations		
Unidirection	al reinforcement			
Membrane r	einforcement in o	ne layer		
- Minimum reinf	orcement			
© 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							
General Bars	Wire fabric reinf.	Con	struction	al reinf.	Shapes		
Bottom reinforce	ement			Top rein ⊂Diame	forcement ter	t	
Direction X Direction Y	12 12	• •	>>	Direct Direct	ion X ion Y	12 12	•
Spacing (cm)	< 10.0 7 10.0		<<	⊂ Spacir ☑ Dir ☑ Dir	ng (cm) rection X rection Y	7. 7.	0
Preferred reinfor	Preferred reinforcement spacing						
Maximum	40.0	cm		Minimun	n	3.0	cm
Direction Y	45.0	cm		Minimun	n	3.0] cm

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.

💳 Plate and Shell Reinforcement 📃 🔲 🔀							
Punchir	Punching						
Verific	ation points			Point grouping			
Name	: S1	30x30		List:	-		
			>>	New	Delete		
1	ew	Delete		Maximum punching forc	e (kN)		
Туре	inte	rnal	•	160.00			
Positi x = [Position (m) x = 2.0000 y = 2.0000 Node number: 2						
Punc	hing:	from t	oottom	▼			
H	ead			- Dimensions (cm	1		
	ype) rootongula		×	a= 30.0			
) rectariguia	11	H	b = 30.0			
circular h = 0.0							
	Qadm	Q	u (m)	Reinforcement	Qadm / Q		
	(KN)	(KN)	(m)	(m), (cm2) / n x ¢			
S1	429.69	162.85	3.8264		2.64 > 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 tounderstand the possible differences for different NADs.

Code	γ _c	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:	ρ _x =ρ _y =0.0077	

MATERIAL:

Concrete: C20/25

FORCES IN THE NODE:

Vertical force: N = 192 kN Moments: $M_x = 24$ kN $M_y = 40$ kN

CALCULATION OF PUNCHING CAPACITY:

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

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

$$d = 0.209m$$

$$\rho = 0.0077$$

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

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

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

$$f_{ctk} = 1.5MPa$$

$$\gamma_{c} = 1.5$$

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

(2.38) (ρc = 2500 kg/m³)

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

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

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

$$A_u = 0.426m^2$$

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

٦

			ca	Manual calculation		obot
Pune	Punching capacity			211 kN		l kN
F Plate ar Punching	nd Shell R	einforcer	ment			<u> </u>
Verifical	tion points			- Point groupin	g	
Name:	S1 3	30x30		List:		•
				New		Delete
Ne	w	Delete		Maximum pu	Inching force	e (kN)
	0.00					
x = 0.1	n (m) 0000 y	= 0.000	10	Node numb	oer:	
Punchi	ng:	from t	ор	*		
Hea	ıd			Dim	ensions (cm	<u>) </u>
	pe rectangula	r	× –		a= 0.0	
				l.	b = 0.0	
	Qadm (kN)	Q (kN)	u (m)	Reinforce (m), (cm2	ement)/nx≬	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 eccentricaly applied support reaction

Example based on: Manual calculation

DESCRIPTION OF THE EXAMPLE:

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

GEOMETRY:

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

REINFORCEMENT:

reinforcement area:	A _x =A _y =16.08	[cm ²]
reinforcement ratio:	ρ _x =ρ _y =0.0077	

MATERIAL:

Concrete:	C20/25
-----------	--------

FORCES IN THE NODE:

Vertical reaction:	V = 192 kN
Moments:	M _x = 24 kN
	$M_{v} = 40 \text{ kN}$

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 $\frac{c_x}{c_y} = 1.67$

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

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

$$v_{Ed} = \beta \frac{V_{Ed}}{ud} = 387kPa$$
$$Q = v_{Ed} \cdot A_u = 342kN$$
$$A_u = ud = 0.883m^2$$

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).

	с	Manual calculation		Robot	
Punching force		342 kN	34	45 kN	
🛲 Plate and Shell Reinforcem	nent			_ 0 x	
Punching					
Verification points		 Point grouping 			
Name: S1 50x30	\rightarrow	List:		•	
	>>	New		Delete	
New Delete		Maximum pun	ching force	e (kN)	
Type: unknown	-	0.00			
x = 0.0000 y = 0.0000)	Node numbe	r:		
Punching: from to	Ρ	*			
Head		⊂ Dime	nsions (cm	J	
Туре	67 <u>d</u>		= 0.0		
rectangular	T	Ль ь	= 0.0		
Circular		h	= 0.0		
Qadm Q (kN) (kN)	u (m)	Reinforcen (m), (cm2)	nent ∕nx≬	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:

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

Code	Punching capacity
EN 1992-1-1:2004	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 are based on the last edition with the corrections EN1992-1-1:2004 AC:2008. However, the last correction does not introduce any changes within the range of the calculations presented in the examples.

DESCRIPTION OF THE EXAMPLE:

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

GEOMETRY:

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

REINFORCEMENT:

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

MATERIAL:

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

LOADS:

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

IMPORTANT STEPS:

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

NOTE:

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

Having calculated the strucutre 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 Thio	:kness		
Homogene	ous		
Label:	h240	Color:	Auto 👻
🔍 🔘 Ca	nstant	Th = 24.	0 (cm)
🔘 Va	riable along a line		
🔘 Va	riable on a plane	r .	
	Point coord	linates	Thicknesses (cm)
P1:	0.0000; 0.0000;	0.0000	0.0
P2:	0.0000; 0.0000;	0.0000	0.0
P3:	0.0000; 0.0000;	0.0000	0.0
Ri mi	eduction of the oment of inertia	1.00	×lg >>>
			1
	Parameters of	foundation	elasticity
Material:	Parameters of	foundation C20/	elasticity

Fig. 4.1. Slab thickness dialog

Job Preferences				? <mark>- X-</mark>
🚅 层 🗙 🔆 🛛 DEFAL	ILTS			•
	• 8 •	F		
- Steel and timber s	Database	Database Name	Database Description	
Standard loads Standard loads Building soils Anchor bolts Reinforcing bars Wire fabrics Wire fabrics The sign codes The sign codes The sign codes The sign codes	EN 1992-1-1 EC2 - ICEL EC2 - ITALI NEN-EN PN 2002_#			4
🚔 <u>O</u> pen default pa	rameters			
Save current paramet	ers as default		Cancel	Help

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

<mark>F</mark> EN 1992-1-1	:2004 AC:2008	Reinforcement	t Para 🗖 🔍	×
General Mate	erials SLS Para	meters Reinfo	rcement	
-Bar dimension	ns			
d1:	12 🗸	d2:	12 🔻	
d1':	12 -	d2':	12 -	
Cover (cm)				51
c1:	1.8	c2:	1.8	
c1':	0.0	c2 ':	0.0	
		Deviations		
Unidirection	al reinforcement			
Membrane r	einforcement in a	one layer		
- Minimum reinl	forcement			
© None				
For FE for which reinforcement As>0				
For the whole panel				
	Note	Add	Close Help	

Fig. 4.3 Definition of covers of reinforcement

Reinforcement Pattern - EN 1992-1-1:2004 AC:2008				
General Bars Wire fat	bric reinf. Constructio	onal reinf. Shapes		
Bottom reinforcement		Top reinforcement		ОК
Diameter		Diameter		Cancel
Direction X A	uto 🔻 >	Direction X	12 🔻	
Direction Y Au	uto 🔻	Direction Y	12 🔻	Help
Spacing (cm)		Spacing (cm)		
Direction X	10.0	Direction X	6.0	Save Ac
Direction Y	10.0	Direction Y	6.0	
				Delete
Preferred reinforcement s	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 eual 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

📻 Plate ar	nd Shell R	einforcer	ment						
Punching									
Verificat	Verification points Point grouping								
Name:	S1 4	S1 40x40 > List:			-				
			>>	>> New Delete					
Ne	w	Delete	Maximum punching force (kN)						
Туре:	unki	nown	-	0.00					
Position x = 0.0	n (m) 2000 y	= 0.000	0	Node number:					
Punchi	ng:	from t	op	Ŧ					
Hea Typ	Head Type Dimensions (cm) Image: state								
	Qadm (kN)	Q (kN)	u (m)	Reinforcement (m), (cm2) / n x ¢	Qadm / Q				
S1	664.67	664.67	4.2389	L1=0.1050 L2=0.1050 A=4.14 / 15 ¢6 L1=0.2428 L2=0.2428 A=4.14 / 15 ¢6	1.00 > 1				
				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	$A=4.14 \text{ cm}^2$
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.

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			? ×
Canadian Constraints of the second se	TANDARD Begional settings: Working language: Printout language:	Eurocode	
Update Preferences o	n 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>		
Units and Formats - Units and Formats - Forces - Other Unit Edition	RD <u>Steel/Aluminum structures:</u>	EN 1993-1:2005 EN 1993-1-8:2005	•		
Materials Databases Design codes Loads Structure Analysis Work Parameters	<u>I</u> imber structures: <u>B</u> C structures: <u>G</u> eotechnical:	PN-8-03150 PN-8-03264 (2002) PN-81/B-03020 <u>M</u> ore codes	• • • • •		
🙀 <u>O</u> pen defaul	Cpen default parameters				
Save current parameters as default OK Cancel Help					

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:*

Codes:			2	Current codes:	
Steel / aluminum		•		Set as current	
Code	Country]	Code	
AL76	France	-		EN 1993-1:2005	
ANSI/AISC 360-05	USA	-		PN-90/B-03200	
ASD:1989 Ed.9th	USA				
Add80	France				
BS - EN 1993-1-1:2005	UK EC3				
BS 5950:2000	UK				
BS5950	UK				
BSK99	Sweden				
CAN/CSA-S16-01 + Supp. No.1 (2005)	Canada	-			
		- • ·		4	

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 ASD:1389 Ed.9th Add80	USA France		2	Code BS-EN 1993-1:2005/NA:2008/AC: DS/EN 1993-1:2005/DK NA:2007/ EN 1993-1:2005/DK NA:2007/ EN 1993-1:1992 NAD Belgium ENV 1993-1:1992 NAD Germany ENV 1993-1:1992 NAD Germany ENV 1993-1:1992 NAD Germany		
BS-EN 1993-1:2005/NA:2008/AC:2009	UK EC3			NS-EN 1993-1:2005/NA:2007/AC.:		
R20200	UK	+		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.

Reg 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		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			
PN-B-03150	Poland			
PN-EN 1995-1:2005/A1:2008	Poland EC5			
		•	< III	•
OK Cancel			<u>H</u> elp	D I

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-defined type of member (e.g. "timber beam") may be initially opened.

Definitions -EN 1995-1:2004/A1:2008 Members Groups 11 Number: • N<u>e</u>w Basic data <u>B</u>ar list: Parameters klasyczna stężona Name: ✓ Member type: klas 1,2 n góra 🔻 C. <u>G</u>roup: Timber Member Timber Column ΟK Save 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 - Paramet	ers - EN 1995-1:2004/A1:2008	
Mem <u>b</u> er type: Timber Beam		Save
Buckling (Y axis) Member length ly: <u>R</u> eal	Buckling (Zaxis) Member length Iz: Real	Close
Coefficient	Coefficient	<u>S</u> ervice
Buckling length coefficient Y:	Buckling lengt <u>h</u> coefficient Z:	<u>M</u> ore
1,00	1,00	<u>O</u> ther
Lateral buckling parameters		<u> </u>
Lat. buckling type:	Lateral buckling length coeff.	
Load level:	ld = lo ld = lo	
	eated as solid	
Method of critical stress determina	ation - 6.3.3 :	Note
Classic - formula [6.31]		
Eor rectangular sections - form	nula (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	×	
₹11 ×	Beam with pinned supports Beam with fixed supports Cantilever Without lateral buckling	OK Cancel Help

select the appropriate Load level icon

💋 Load Level	×
	OK Cancel Help

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

Member Definition - A	Additional Parar	meters X
Load parameters Load type:		OK Cancel
Section parameters Anet/Agross ratio	1,00	Help
Additional conditions for re Unidirectional bending	ound sections g	

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

💋 Load Type	×
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	Buckling Length Coet	fficient 💌
	ld = 2lo	OK Cancel
	ld = lo	Help
	ld = 0.5 lo	
	<u>l</u> d = 1,00 lo	
X		
Ŧ	Intermediate bracings	

The second one opens BUCKING TYPE DIAGRAMS dialog:

Buckling Type Diagrams	X
$\begin{bmatrix} 1 & 1 & 1 & 1 \\ 10 & 0.5 & 0.7 & 2.0 \\ \hline $	OK Cancel
<u>C</u> omplex member Batten type: packs	Help
Connection: glued	
× L O real O relative	

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

🗲 Internal bracings		X
Z Lateral	bucklin	ALSO M
Test for member: 1 Belka stęzona Buckling Y Buckling Z Lateral buckling-upper flange Lateral buckling-upper flange Coordinates of the existing bracings Coordinates buckling bracings	al buckling	Hower flange
Detine 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.: I Belka stęzona
Buckling coefficients of component segments		For load case:
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.

Member Definition - Parameters - EN 1995-1:2004/A1:200	8		
Mem <u>b</u> er type: klas 1,2m góra st	Save		
Buckling (Y axis) Member length ly: Beal Coefficient Buckling (Z axis) Member length lz: Real 1,00 Coefficient 1,00	Close		
Buckling length coefficient Y: 1,00 1,00 Buckling length coefficient Z: 1,00 Buckling length coefficient Z:	<u>Service</u>		
Lateral buckling parameters	<u> </u>		
Load level:			
Double sections treated as solid Method of critical stress determination - 6.3.3 : Classic - formula [6.31] For rectangular sections - formula [6.32]	Note		
S <u>e</u> rvice class: 1	Help		
	Definitions -EN	1995-1:2004/A1:2008	
nber of the member must be	Members Groups	3 1 √	New
<u>lember type</u> .	Basic data Bar list:	1	
t is very important when verifying different member types	<u>N</u> ame:	Belka stęzona	Parameters

ΟK

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

In the CALCULATIONS dialog set the

-> Loads cases - list of chosen loads

-> Verification options - list of verified members,

	.2004/A1.2000	
• <u>M</u> ember verification:	1to6 11to16	List
Code group verification:		List
Code group <u>d</u> esign:		List
Optimization	Options	;
Loads		Limit state
C <u>a</u> ses: 8	List	🔽 <u>U</u> ltimate
Calculation archive Save calculation <u>r</u> esults	List	Serviceability
OK	uration Calcula	ations Help

<u>S</u>ave

Help

following:

-> Limit state ->Configuration. Before doing calculations you have to remember to specify appropriate duration for loads in the CALCULATIONS dialog:

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

Configuration		Ru Load Case Classification - Duration
Calculation points	ОК	Case list:
Calculation points Number of points: Characteristic points Characteristic points Calculation parameters Efficiency ratio: Maximum slenderness: 210,00 Maximum slenderness: 210,00 Maximum slenderness: 210,00 Components of complex bars are not taken into account Calculations: fire impact Load case classification - duration Exclude internal forces from calculations Units of results Code Robot Camber Take the deflections from the	OK Cancel Help	Class Classification - Duration Case list: Number Name 4 wiatr 5 s nierzutowane 8 wypadkowe obc • 8 • • •
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:

Reg Load Case Classification - De	uration					×
Case list:						
Number Name		Class		Number	Name 🔶	ОК
			4	4	wiatr	Cancel
			4	5	s nierzut 📄	Cancor
			4	8	wypadkc 📮	
<	• <<	•	111		4	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.

DEN 1995-1:2004/A1:2008 -	Member Verification	(ULS) 1to6	11to16				
Results Messages							Calc. Note Close
Member	Section	Material	Lay	Laz	Ratio	Case	Help
1 Belka stęzona	BALK 75x200	C16	83.14	221.71	0.89	8 wypadkowe obc	Devie
2 Belka obc. górą	BALK 75x200	C16	83.14	221.71	0.89	8 wypadkowe obc	
3 Belka wolnopodp	BALK 75x200	C16	83.14	221.71	0.89	8 wypadkowe obc	Analysis Map
4 Belka stęzona	BALK 75x200	C16	83.14	221.71	0.89	8 wypadkowe obc	Calculation points
5 Belka obc. górą	BALK 75x200	C16	83.14	46.19	0.89	8 wypadkowe obc	Division: n = 3
6 Belka wolnopodp	BALK 75x200	C16	83.14	221.71	0.89	8 wypadkowe obc	Extremes: none
11 klasyczna stęzona	BALK 75x200	C16	121.24	323.32	1.89	8 wypadkowe obc	Additional: none
12 klas obc. górą	BALK 75x200	C16	121.24	323.32	1.89	8 wypadkowe obc	
13 klasycz obcdół	BALK 75x200	C16	121.24	323.32	1.89	8 wypadkowe obc	
14 uproszcz stężona	BALK 75x200	C16	121.24	323.32	1.89	8 wypadkowe obc	
15 Belka obc. górą	BALK 75x200	C16	121.24	46.19	1.89	8 wypadkowe obc	
16 uproszcz obcdól	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

PRESULTS - Code - EN 1995-1:2004/A1:2008		
Bar: 1 Belka stęzona Point / Coordinate: 2 Load case: 8	Section OK	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 MPa	Lambda_rel m = 0.48 k.crit = 1.00	Help
BUCKLING Y	BUCKLING Z	
\mathbf{X}	×	
RESULTS Sig_m.y.d/f m.y.d + km*Sig_m.z.d/f m.z.d = 6.38/11.08 + 0. Sig_m.y.d/(kcrit*f m.y.d) = 6.38/(1.00*11.08) = 0.58 < 1.00	70°5.68/12.72 = 0.89 <1.00 (6.11) (6.33)	

Detailed results tab

ALK 75x200	<u>A</u> uto ▼	Bar: 1 Point / Coo Load case:	Belka stęzona Ser rdinate: 2 / x = 0.50 L = 2.40 m 8 wypadkowe obc	ction OK	。 。	OK
implified results	Detailed results	;				<u>C</u> han <u>c</u>
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]		Eorce
		Fac	tors and additional parameters			Detaile
kh	1.15		Scale coefficient	[3.2/3.3/3.4]		
kh_y	1.00		Scale coefficient	[3.2/3.3/3.4]		
kh_z	1.15		Scale coefficient	[3.2/3.3/3.4]		
kl	1.00		Reduction factor depending on member length	[3.4.(4)]		
kmod	0.90		Modification factor depending on time of load action	[3.1.3]		Laic. 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	itical 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		

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

RESULTS:

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

TIMBER STRUCTURE CALCULATIONS

CODE: EN 1995-1:200 ANALYSIS TYPE: Me	04/A1:2008 mber Verification			
CODE GROUP: MEMBER: 1 Belka stę	zona POINT: 2	COORDINA	TE: $x = 0.50 L = 2.40 m$	
LOADS: Governing Load Case: 8	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	
SECTION PARA	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= 3.1 Sig_m,z,d = MZ/Wz= 1.0	9/500.00 = 6.38 MPa 7/187.49 = 5.68 MPa	ALLOWABLE STRESSES f m,y,d = 11.08 MPa f m,z,d = 12.72 MPa		
Factors and additiona $km = 0.70$ $kh = 1.1$	I parameters 5 kmod = 0.90 k	Ksys = 1.00		
LATERAL BUCK 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:		
VERIFICATION FORM Sig_m,y,d/f m,y,d + km*S Sig_m,y,d/(kcrit*f m,y,d)	ULAS: Sig_m,z,d/f m,z,d = 6.38/11.0 = 6.38/(1.00*11.08) = 0.58 <	08 + 0.70*5.68/12.72 = 0 < 1.00 (6.33)	.89 < 1.00 (6.11)	

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 / Load case: 8 v	Incorrect section	OK
BALK 50x115	sults		<u>C</u> hange
BALK 50x125 BALK 50x140 BALK 50x150 BALK 50x175 BALK 50x200 BALK 50x225 BALK 50x225 BALK 50x250 BALK 53x140	8.29 MPa 5.57 MPa	ALLOWABLE STRESSES f m.y.d = 12.01 MPa f m.z.d = 13.80 MPa	<u>Forces</u>
BALK 63x150 BALK 63x160 BALK 63x175 BALK 63x200 BALK 63x225 BALK 63x250 BALK 75x150 BALK 75x160 DALK 75x160	VAL PARAMETERS 1.25 kmod = 0.90	Ksys = 1.00	Calc. Note
BALK 75x200	lef = 1.28 m	lambda relm = 0.48	
BALK 75x225 BALK 75x250	Sig cr = 68.56 MPa	k crit = 1.00	
BALK 100x200 BALK 100x225 BALK 100x250			нар
RESULTS Sig_m.y.d/1 m.y.d + km Sig_m.y.d/(kcrit*f m.y.d	'Si <u>a_</u> m,z,d/f m,z,d = 38,29/12.01 + 0) = 38,29/(1.00°12.01) = 3.19 > 1.00	.70°25.57/13.80 = 4.48 > 1.00 (6.11) (6.33)	

Simplified results	8 wypadkowe obc
CALCULATION STRESSES	ALLOWABLE STRESSES
Sig_m,y,d = 3.19/531.56 = 6.00 MPa Sig_m,z,d = 1.07/148.83 = 7.16 MPa	fm.y.d = 11.08 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 = 1.53 m Sig_cr = 44.3	Lambda_rel m = 0.60 37 MPa k crit = 1.00
RESULTS km*Sig_m.y.d/f m.y.d + Sig_m.z.d/f m.z.d = 0.70°	*6.00/11.08 + 7.16/13.18 = 0.92 < 1.00 (6.12)

The results for the newly selected section are presented below.

TIMBER	STRUCTURE CAL	LCULAT	IONS for BALL	K 63x225
CODE: EN 1995-1:20 ANALYSIS TYPE: M	004/A1:2008 ember Verification			
CODE GROUP: MEMBER: 1 Belka ste	zona POIN	T: 2 C	OORDINATE: $x = 0.50$	0 L = 2.40 m
LOADS: Governing Load Case: 8	3 wypa dkowe obc			
MATERIAL C16 gM = 1.30 f v,k = 1.80 MPa E 0,05 = 5400.00 MPa	f m,0,k = 16.00 MP f t,90,k = 0.50 MPa G moyen = 500.00 N	°a ⁄IPa	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 MF Beta c = 1.00
bt=22.5 cm	ARAMETERS: BALK 6	j3x225		
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. Sig_m,z,d = MZ/Wz= 1.0	19/531.56 = 6.00 MPa)7/148.83 = 7.16 Mpa		ALLOV f m,y,c f m,z,d	VABLE STRESSES 1= 11.08 MPa = 13.18 MPa
Factors and additiona km = 0.70	al parameters kh = 1.19 k	mod = 0.90	Ksys = 1.00	
LATERAL BU lef = 1.53 m Sig_cr = 44.37 MPa	CKLING PARAMETER Lambda_re k crit = 1.0	S: l m = 0.60 0		
BUCKLING PARAME About Y axis:	TERS:		X	About Z axis:
VERIFICATION FORM km*Sig_m,y,d/f m,y,d + Sig_m,y,d/(kcrit*f m,y,d)	IULAS: Sig_m,z,d/f m,z,d = 0.70*6 = 6.00/(1.00*11.08) = 0.5	6.00/11.08 + 54 < 1.00 (6	7.16/13.18 = 0.92 < 1. 5.33)	00 (6.12)

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.



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 1	995-1:2004/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	ers - EN 1995-1:2004/A1:2008	×			
Member type: Timber Column		Save			
Buckling (Yaxis) Member length ly:	Buckling (Z axis) Member length lz:	Close			
 <u>Heal</u> <u>Co</u>efficient 	 Real 1,00 Coef<u>ficient</u> 	<u>S</u> ervice			
Buckling length coefficient Y:	Buckling length coefficient Z:	More			
1,00	1,00	0ther			
Lateral buckling parameters	<u> </u>				
Lat. buckling type:	Lateral buckling length coeff.				
Load level:	ld = lo ld = lo				
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	Buckling Type	×
X	Beam with pinned supports Beam with fixed supports Cantilever Without lateral buckling	OK Cancel Help

select appropriate Load level icon

Doad Level	×
	OK Cancel Help

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

Member Definition - Ac	dditional Paran	neters X
Load parameters		OK Cancel
Section parameters Anet/Agross ratio	1,00	Help
Additional conditions for rou	und sections	

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

💋 Load Type	×
Moment at the end Uniform load Concentrated 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,

💋 Lateral	Buckling Length Coef	ficient 💌
	ld = 2lo	OK Cancel
	ld = lo	Help
	ld = 0.5 lo	
	<u>l</u> d = 1,00 lo	
X		
Ŧ	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			
	buckli	ng-lower flange	
Buckling Y Buckling Z Lateral buckling-upper flange Latera	al buckling	lower flange	
Coordinates of the existing bracings Profile execution to a state of the existing bracings		Automatic detection of brac	angs
Denne manually cooldinates of the existing bracings	m	Add bracings at points v Zero	where bending moments equal
I geal		Bracing detection preview	
		For member no.:	3 Nc+My bez wyb z-z 🔹 👻
		For load case:	1 STA1 👻
Buckling coefficients of component segments			m
OK Cancel			Help

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

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



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

→ it is very important when verifying different member types

💋 Defir	itions -EN 1995	-1:2004/A1:2008	
Memb	ers Groups		
N <u>u</u> n	nber:	3 🗸	New
Bas	ic data		
<u>B</u> ar	list:	3	
<u>N</u> an	ne:	Nc+My bez wyb z-z	Parameters
C. <u>G</u>	įroup:	✓ Member type:	Słup śc stęż z 👻
	IK	<u>S</u> ave	

In 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:						
Code group verification:						
Code group <u>d</u> esign:	List					
	Options					
Loads	Limit state					
	Ultimate					
Save calculation results	List Serviceability					
OK Configur	ation Calculations Help					

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		23	1	Reg Load Cas	e Classification - Dura	tion				X
Calculation points		ОК		Case list:		_				_
Number of points:	3 🗧	Cancel		Number	Name	>	Class	Number	Name	OK
Characteristic points	Options			1	STA1	\rightarrow				Cancel
- Calculation parameters		Help		2	widu					
Efficiency ratio:	1,00 l			•		<<	•	III	F	Help
Maximum slenderness:	210,00			Load class ac	cording to duration:					
Components of complex not taken into account	bars are			1. Permanent		-				
Calculations: fire impact				2. Long-term			_			
Load case classification	- duration			3. Medium-te 4. Short-term 5. Instantane	m ous					
Exclude internal forces from	calculations									
Units of results Code Rol	bot		Ŀ							
Camber										
following case into consid	n (ne deration:									
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:

Reg Load Case Classification - Duration								
Case list:								
Number Name		Class	Number	Name	ОК			
	>>	1 5	1 2	STA1 wiatr	Cancel			
•	• <<	•	1		• 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.

6	EN 1995-1:2004/A1:20								
	Results Messages	Calc. Note Close							
	Member		Section	Material	Lay	Laz	Ratio	Case	Help
	1 Nc	ок	PROST_44x145	C16	57.34	188.95	0.36	1 STA1	Delia
	2 My	OK	PROST_44x145	C16	57.34	188.95	0.79	2 wiatr	Hatio
	3 Nc+My bez wyb z-z	ОК	PROST_44x145	C16	57.34	188.95	0.91	3 KOMB1	Analysis Map
	4 bez zwich	ок	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	0K	PROST_44x145	C16	57.34	47.24	0.91	3 KOMB1	Additional: none

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:	Bar: 3 Nc+My bez wy Point / Coordinate: 2 / Load case: 3 1	yb z-z / x = 0.50 L = 1.20 m KOMB1 (1+2)*1.00	
Simplified results Detailed results	1		Change
CALCULATION STRESSES Sig_c.0.d = 12.00/63.80 = 1.88 Sig_m.y.d = 1.50/154.18 = 9.73	MPa MPa	ALLOWABLE STRESSES f c.0,d = 14.38 MPa f m.y,d = 13.63 MPa	Eorces Det <u>a</u> iled
FACTORS AND ADDITIONAL km = 0.70 kh = 1.20 LATERAL BUCKLING	PARAMETERS 8 kmod = 1.10 lef = 2.45 m Sig or = 20.79 MPa	Ksys = 1.00 Lambda_rel m = 0.88 k.cnt = 0.90	Calc. Note Parameters
BUCKLING Y LY = 2.40 m LFY = 2.40 m Lambda Y = 57.34	Lambda_relY = 1.02 ky = 1.10 kcy = 0.67		Help
ncoucto Sig_c,0,d/(kc,y*f c,0,d) + Sig Sig_m,y,d/(kcrit*f m,y,d) = 9.7:	m.y.d/f m.y.d = 1.88/(0.67*14 3/(0.90*13.63) = 0.79 < 1.00	1.38) + 9.73/13.63 = 0.91 < 1.00 (6.23) (6.33)	

Detailed results tab

	<u>A</u> uto	Bar: 3 Point / Coo Load case:) Nc+My bez wyb z-z dinate: 2 / x = 0.50 L = 1.20 m 3 KOMB1 (1+2)*1.00	Section OK	○○○	<u>ОК</u>
mplified results	Detailed results	:				<u>C</u> han
Symbol	Value	Unit	Symbol description	Section	^	
km	0.70		Interaction factor due to bending	[6.1.6.(2)]	1	
Ksys	1.00		System coefficient	[6.7]]	
			Buckling parameters]	Eorce
About the Y a	axis of cross-s	ection			-	Det <u>a</u> il
LY	2.40	m	Member length	[6.3.2]	1	
LFY	2.40	m	Buckling length	[6.3.2]	1	
Lambda Y	57.34		Member slenderness	[6.3.2]	1	
Sig c,crit,y	16.21	MPa	Critical stress (buckling)	[6.3.2.(1)]	1	
Lambda_rel	1.02		Relative slenderness (buckling)	[6.3.2.(1)]	1	Lalc. N
ky	1.10		Slenderness factor	[6.3.2.(3)]	1	Parame
kcy	0.67		Reduction factor due to compression	[6.3.2.(3)]	1	
		Paran	neters of lateral buckling analysis	,	1	Help
Method of cr	itical stress d	eterminatio	1 - Classic - formula (6.31)		1	
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)]	1	
k crit	0.90		Lateral buckling factor	[6.3.3.(4)]	1 =	
			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 1995-1:200 ANALYSIS TYPE: Mer	04/A1:2008 nber Verification					
CODE GROUP: MEMBER: 3 Nc+My be	ez wyb z-z PC	DINT: 2 COORDIN	ATE: $x = 0.50 L = 1.20 m$			
LOADS: Governing Load Case: 3	KOMB1 (1+2)*1.00					
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: 2	f c,0,k = 17.00 MPa E 0,moyen = 8000.00 MPa Beta c = 0.20			
bt=14.5 cm	METERS: PROST_44x	.145				
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/ Sig_m,y,d = MY/Wy= 1.50	63.80 = 1.88 MPa 0/154.18 = 9.73 MPa	ALLOWABLE STRESSES f c,0,d = 14.38 MPa f m,y,d = 13.63 MPa				
Factors and additional $km = 0.70$ $kh = 1.28$	parameters 8 kmod = 1.10	Ksys = 1.00				
LATERAL BUCK lef = 2.45 m Sig_cr = 20.79 MPa	LING PARAMETERS: Lambda_rel m = 0.88 k crit = 0.90					
BUCKLING PARAMET \downarrow_{10} About Y axis: LY = 2.40 m Lambda_rel Y = 1.02 LFY = 2.40 m	ERS: Lambda Y = 57.34 ky = 1.10 kcy = 0.67	About Z axis:				
VERIFICATION FORMU Sig_c,0,d/(kc,y*f c,0,d) + Sig_m,y,d/(kcrit*f m,y,d) =	JLAS: Sig_m,y,d/f m,y,d = 1.88/(0 = 9.73/(0.90*13.63) = 0.79	0.67*14.38) + 9.73/13.63 0 < 1.00 (6.33)	= 0.91 < 1.00 (6.23)			

Section OK !!!

COMPARISON:

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

verifications parameters, interaction expression		Robot	Handbook
$\begin{array}{l} \lambda y \ \ - \ member \ slenderness \\ ky \ \ - \ slenderness \ factor \\ kcy \ \ - \ reduction \ factor \ due \ to \ compression \\ k_{mod} \\ f \ c,o,d \ \ \ design \ compression \ strength \\ f \ m,y,d \ \ \ design \ bending \ strength \ due \ to \ My \\ \sigma \ c,o,d \ \ \ design \ bending \ stress \ due \ to \ My \end{array}$	[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.