# Autodesk<sup>®</sup> Robot<sup>™</sup> Structural Analysis Professional

## VERIFICATION MANUAL FOR EU CODES

March 2014

© 2014 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 Professional, 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.

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.

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 3 - COMBINED COMPRESSION AND BENDING.	
VERIFICATION EXAMPLE 4 - BENDING WITH LATERAL BUCKLING	
VERIFICATION EXAMPLE 5 - COMBINED BI-AXIAL BENDING AND COMPRESSION	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	23
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	33
VERIFICATION EXAMPLE 6 - DEFLECTION OF SIMPLY SUPPORTED BEAM WITH RECTANGULAR SECTION	38
LITERATURE	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
LITERATURE	
3. EUROCODE 2 EN 1992-1-1:2004 AC:2008 - RC SLABS (PUNCHING)	49
VERIFICATION PROBLEM 1 - PUNCHING CAPACITY OF SLAB WITHOUT SHEAR REINFORCEMENT	50
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	
TIMBER	
1. EUROCODE 5: DESIGN OF TIMBER STRUCTURES; EN 1995-1:2004/A1:2008	67
GENERAL REMARKS	68
VERIFICATION PROBLEM 1 BENDING ABOUT TWO MAIN AXES WITH LATERAL BUCKLING	70
VERIFICATION PROBLEM 2 COMBINED COMPRESSION AND BENDING ABOUT ONE MAIN AXIS	80

## INTRODUCTION

This verification manual contains numerical examples for structures prepared and originally calculated by **Autodesk Robot Structural Analysis Professional version 2013**. The comparison of results is still valid for the next versions.

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

Each example contains the following parts:

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

## STEEL

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

## VERIFICATION EXAMPLE 1 - Axial compression

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

#### TITLE:

Axial compression (Example 6.2 page 44).

#### SPECIFICATION:

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



#### SOLUTION:

Define a new type of member. For 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	Buckling (Y axis) Member length ly: Member length lz:	Close
Basic data	Con the Con th	
Barlist: 1	© Coefficient 1,00 Coefficient 1,00	
Name: Bar 1 Parameters	Buckling length coeff. Y: Buckling length coeff. Z:	
C. Group: Member type: Column1	Non-sway Non-sway	
	Buckling curve Y auto V Bucking curve Z auto V	
OK <u>S</u> ave Help	Elexural-torsional buckling of monosymmetric sections	
	Lateral buckling parameters	Harr
<b>Buckling Diagrams</b>	Lateral buckling Lateral buckling length coefficient	<u>M</u> ore
	Load level:	
1.0 0.5 0.7 2.0 · Cancel	Ler = lo Ler = lo	
	C <u>G</u> eneral method [6,3,2,2] Lambda LT,0 = 0,40	
	Detailed method [6:3.2.3]     Beta = 0,75     Simplified method for beams with	
	$\label{eq:simplified method for beams with lateral restraints [\underline{6}:3,2,4]  k\underline{f} = 1,10$	
C. Surger alterative	Additional sets of member parameters	
C Sway structure Non-sway structure	<u>L</u> imit deflections and displacements: <u>Service</u>	
S Touring automo	Complex sections:	
	Ihin-walled sections:	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>Member verification:</u> <u>1</u> <u>List</u>
C Code group verification:
O Code group design:
Optimization     Options
Loads
Cases: 1 List Vitimate
Calculation archive
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 Messages									Close
	Member	Section	Material	Lay	Laz	Ratio	Case			Help
	1 Bar 1	UC 254x254x7	S355	36.13	61.74	1.00	1 STA1		⊢ Ratio ———	
									Analysis	Map
									Calculation p Division:	n = 3
									Extremes:	none
L									Additional:	none

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

🚰 RESULTS - Code - EN 1993-1:2005							_ 🗆 🗙
Auto         Bar         1 Bar 1           UC 254x254x73         ▼         Section 0K         ●	<u>(СОК</u> )	UC 254×254×7	<u>Auto</u>	Bar: Point / Cor Load case:			<u>( 0K )</u>
Simplified results Detailed results	Change	Simplified result:	5 Detailed results	1			Change
F0RCS N.E.4 = 305 M N.R.4 = 305 M N.R.4 = 305 M		Symbol	Values	Unit Cross-	Symbol description Section section properties: UC 254x254x73	n 🔺	
	Eorces	Ax	93.100	cm2	Cross-section area		Eorces
	Detailed	Ay	75.883	cm2	Shear area - Y-axis		Detailed
Class of section = 2		Az	25.792	cm2	Shear area - Z-axis		
LATERAL BUCKLING		bx	57.600	cm4	Torsional constant		
×LT = 1.00		ly	11410.000	cm4	Moment of inertia of a section about the Y-axis		
		Iz	3908.000	cm4	Moment of inertia of a section about the Z-axis		
BUCKLING Y BUCKLING Z	Calc. Note	Wpły	990.000	cm3	Plastic section modulus about the Y (major) axis		Calc. Note
$\mathbf{X}$		Wplz	463.000	cm3	Plastic section modulus about the Z (minor) axis		
	Parameters	h	25.40	cm	Height of cross-section		Parameters
		b	25.40	cm	Width of cross-section		
	Help	tf	1.42	cm	Flange thickness		Help
SECTION CHECK		tw	0.86	cm	Web thickness		
N.Ed/Nc.Rd = 1.00 < 1.00 (6.2.4.(1))		ry	11.07	cm	Radius of gyration - Y-axis		
		rz	6.48	cm	Radius of gyration - Z-axis		
MEMBER STABILITY CHECK					Material:		
Not analyzed		Name			\$355 (\$355)		
		fy	355.00	MPa	Design yield strength of material (3.2)	<u> </u>	

## STEEL DESIGN

	993-1:2005, Eurocode 3: De PE: Member Verification	esign of steel structures.	
CODE GROUP MEMBER: 1 E 0.00 m		1	<b>COORDINATE:</b> $x = 0.00 L =$
LOADS: Governing Load	Case: 1 STA1		
MATERIAL: S355 (S355)	fy = 355.00 MPa		
<b>SECTIO</b> h=25.40 cm b=25.40 cm tw=0.86 cm tf=1.42 cm	DN PARAMETERS: UC 254 gM0=1.00 Ay=75.883 cm2 Iy=11410.000 cm4 Wply=990.000 cm3	gM1=1.00 Az=25.792 cm2 Iz=3908.000 cm4	Ax=93.100 cm2 Ix=57.600 cm4
INTERNAL FOI N,Ed = 3305 kN Nc,Rd = 3305 kN Nb,Rd = 3305 kN			Class of section = 2
	AL BUCKLING PARAMETE	ERS:	
BUCKLING PA	RAMETERS:		
About Y	axis:	About Z ax	is:
<b>VERIFICATION</b> Section strength N,Ed/Nc,Rd = 1.0			

#### Section OK !!!

#### **COMPARISON:**

Resistance, interaction expression	Robot	Handbook
1. design compression resistance of the cross+section N <sub>c.Rd</sub>	3305	3305

## VERIFICATION EXAMPLE 2 - Axial compression with buckling

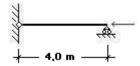
Example taken from Designer's Guide to EN 1993-1-1 L.Gardner and D.A.Nethercot, Thomas Telford Publishing, 2005

#### 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 📃 🖂 🔀	f Member Definition - Parameters - EN 1993-1:2005	×
Members Groups	Member type: Column2	Save
Number:     1     New       Basic data	Buckling (Y axis)       Buckling (Z axis)         Member length ly:       Member length lz:         © <u>C</u> oefficient       Real         © <u>C</u> oefficient       1.00         Buckling length coeff. Y:       Buckling length coeff. Z:         1.00       1.00         Non-sway       Non-sway         Buckling curve Y       auto ▼	Close
Buckling Diagrams $X$ $1$ $1$ $1$ $1$ $0$ $1$ $1$ $1$ $1$ $1$ $1$ $0$ $1$ $1$ $1$ $1$ $1$ $1$ $0$ $1$ $1$ $1$ $1$ $1$ $1$ $1$ $0$ $1$ $1$ $2$ $1$ $2$ $1$ $2$ $0$ $0.8$ $X$ $y$ $y$ $1$	Flexural-torsional buckling of monosymmetric sections         Lateral buckling       Lateral buckling length coefficient         Load level:       Lower flange         Load level:       Lor = lo         Lor = lo       Lor = lo         C general method       [6:3.2.2]         Lambda LT.0 =       0.40         © Detailed method       [6:3.2.3]         Beta =       0.75         Simplified method for beams with lateral restraints [5:3.2.4]       1.10         Additional sets of member parameters       Limit deflections and displacements:         Complex sections:       Complex	More
	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	
Member verification:	1 List
C Code group verification:	List
C Code group <u>d</u> esign:	List
<u>Optimization</u>	Options
Loads	Limit state
Cases: 1	List 🔽 Ultimate
Calculation archive	
Save calculation <u>r</u> esults	
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									( <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		r Ratio	
									Analysis	Map
									Calculation po Division:	ints 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.

5 RESULTS - Code - EN 1993-1:2005	_ 🗆 🗙	🛛 🗾 RESULTS - Co	de - EN 1993-1:	2005			_ 🗆 🗵
Auto         Bar:         1 Bari         Section 0K         Image: CHS 244.5x10         Image: CHS 244.5x10	<u> </u>	CHS 244.5x10	Auto	Bar: Point / Cor Load case			<u>ОК</u>
Simplified results Detailed results	Change	Simplified results	s Detailed results				Dhange
FORCES N.Ed - 15300 kN		Symbol	Values	Unit	Symbol description Section	-	
Nc,Rd = 2026.8 kN Nb,Rd = 1836.5 kN				Cross	s-section properties: CHS 244.5x10		
	Eorces	Ax	73.700	cm2	Cross-section area		Eorces
		Ay	46.919	cm2	Shear area - Y-axis		
Class of section = 1		Az	46.919	cm2	Shear area - Z-axis		
LATERAL BUCKLING		lx .	10150.000	cm4	Torsional constant		
×LT = 1.00		ly	5073.000	cm4	Moment of inertia of a section about the Y-axis		
		Iz	5073.000	cm4	Moment of inertia of a section about the Z-axis		
- BUCKLING Z	Calc. Note	Wply	550.236	cm3	Plastic section modulus about the Y (major) axis		Calc. Note
Ly = 4.00 m Lam_y = 0.56 Lz = 4.00 m Lam_z = 0.56	- cgic. Hote	Wplz	550.236	cm3	Plastic section modulus about the Z (minor) axis		- cgic. Hote
10 Lcr,y = 4.00 m Xy = 0.91 10 Lcr,z = 4.00 m Xz = 0.91	Parameters	h	24.45	cm	Height of cross-section		Parameters
Lamy = 48.21		b	24.45	cm	Width of cross-section		
Lamy = 48.21	Help	tf	1.00	cm	Flange thickness		Help
- SECTION CHECK		tw	1.00	cm	Web thickness		nep
N.E.d/Nic.Rd = 0.80 < 1.00 (6.2.4.(1))		ry	8.30	cm	Radius of gyration - Y-axis		
		rz	8.30	cm	Radius of gyration - Z-axis		
MEMBER STABILITY CHECK Lamy = 48.21 < Lam.max = 210.00 Lamz = 48.21 < Lam.max = 210.00 STABLE					Material:		
N.Ed/Nb.Rd = 0.89 < 1.00 (6.3.1.1.(1))		Name			\$275 (\$275)		
		fy	275.00	MPa	Design yield strength of material (3.2)	-	
	]						

## STEEL DESIGN

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

#### COMPARISON:

Resistance, interaction expression	Robot	Handbook
1. cross-section compression resistance N <sub>c.Rd</sub>	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:         OK       Save	Member Definition - Parameters - EN 1993-1:2005         Member Lype:       Column1         Buckling (Y axis)       Buckling (Z axis)         Member length ly:       Member length lz:         Beal       Coefficient         © Coefficient       1.00         Buckling length coeff. Y:       Buckling length coeff. Z:         1.00       Image: Non-sway         Buckling curve Y       Buckling curve Z
	Elexural-torsional buckling of monosymmetric sections
Buckling Diagrams $K$ I       I <th>Lateral buckling parameters       Lateral buckling length coefficient.         Lateral buckling       Lateral buckling length coefficient.         Load level:       Lpper flange         Lor = lo       Lcr = lo         C General method [6.3.2.2]       Lambda LT,0 =         0       Detailed method [6.3.2.3]       Beta =         0.75       Simplified method for beams with kfi =       1.10         Additional sets of member parameters       Limit deflections and displacements:       Service         Complex sections:       Domplex.       Domplex.</th>	Lateral buckling parameters       Lateral buckling length coefficient.         Lateral buckling       Lateral buckling length coefficient.         Load level:       Lpper flange         Lor = lo       Lcr = lo         C General method [6.3.2.2]       Lambda LT,0 =         0       Detailed method [6.3.2.3]       Beta =         0.75       Simplified method for beams with kfi =       1.10         Additional sets of member parameters       Limit deflections and displacements:       Service         Complex sections:       Domplex.       Domplex.
	Thin-walled sections: Thin-walled Note
	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.

<b>f</b> Calculations - EN 1993-1:2005	X
Verification options	
<u>Member verification:</u> <u>1</u> <u>List</u>	
C Code group verification:	1
C Code group design:	
Optimization     Options	
Loads Limit state	
Cases: 1	
Calculation archive	
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 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		r Ratio			
									Analysis	Map		
									Calculation poi Division: r	nts n = 3		
									Extremes: r	none		
									Additional: r	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.

12 RESULTS - Code - EN 1993-1:2005	_ 🗆 🗵	S 🖉 RESULTS - Code - EN 1993-1:2005	- 🗆 ×
Auto         Bar:         1 Bari         Section 0K         Image: Coordinate:         1 / x = 0.00 L = 0.00 m         Im	<u> </u>		
Simplified results Detailed results	Change	Simplified results Detailed results	nge
F0RCES N,Ed = 1400.0 kN My,Ed = -342.2 kN*m		Symbol Values Unit Symbol description Section	
Nc,Rd = 2937.5 kN My,p,IRd = 525.0 kN*m Nb,Rd = 2937.5 kN My,c,Rd = 525.0 kN*m		Cross-section properties: UB 457x191x98	
My,N,Rd = 342.5 kN*m	Eorces	Ax 125.000 cm2 Cross-section area Eoro	xes 🛛
	Detailed	Ay 78.511 cm2 Shear area - Y-axis Deta	hat
Class of section = 1		Az 55.655 cm2 Shear area - Z-axis	icu
LATERAL BUCKLING		bx 121.000 cm4 Torsional constant	
XLT = 1.00		ly 45730.000 cm4 Moment of inertia of a section about the Y-axis	
		Iz 2347.000 cm4 Moment of inertia of a section about the Z-axis	
FBUCKLING Y	Calc. Note	Wply 2234.000 cm3 Plastic section modulus about the Y (major) axis Calc. 1	Note
$\mathbf{X}$		Wplz 379.000 cm3 Plastic section modulus about the Z (minor) axis	
	Parameters	h 46.74 cm Height of cross-section Param	eters
		b 19.28 cm Width of cross-section	
	Help	tf 1.96 cm Flange thickness Hel	- I
- SECTION CHECK		tw 1.14 cm Web thickness	¢
My,Ed/My,N,Rd = 1.00 < 1.00 (6.2.9.1.(2))		ry 19.13 cm Radius of gyration - Y-axis	
		rz 4.33 cm Radius of gyration - Z-axis	
MEMBER STABILITY CHECK		Material:	
Not analyzed		Name \$235 (\$235 )	
		fy 235.00 MPa Design yield strength of material (3.2)	

## STEEL DESIGN

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

### COMPARISON:

Resistance, interaction expression	Robot	Handbook
1. plastic moment resistance M <sub>pl,y, Rd</sub>	525,0	524,5
2. reduced plastic moment resistance M <sub>N,y,Rd</sub>	342,5	342,2

### VERIFICATION EXAMPLE 4 - Bending with lateral buckling

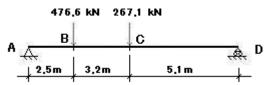
Example taken from Designer's Guide to EN 1993-1-1 L.Gardner and D.A.Nethercot, Thomas Telford Publishing, 2005

#### 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.



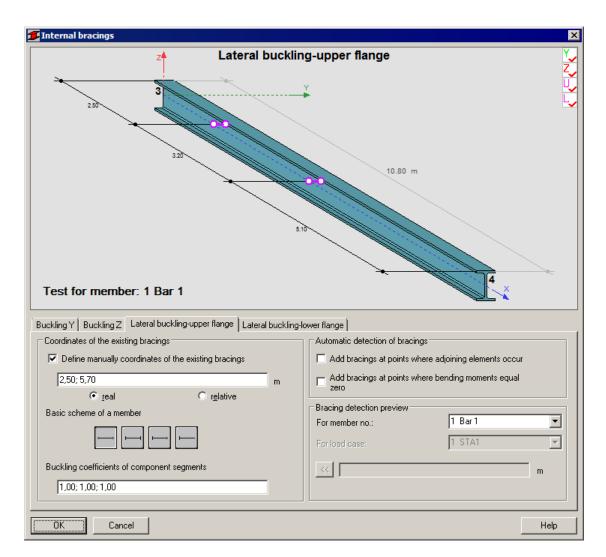
#### SOLUTION:

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	Member type: Beam1 Save
Number:     1     New       Basic data     Basic data       Bar list:     1       Name:     Bar 1       C. Group:     Member type:       OK     Save	Buckling (Y axis)       Close         Member length ly:       Member length lz:         Beal       1,00         Coefficient       1,00         Buckling length coeff. Y:       1,00         1,00       Image: State
Lateral Buckling Length Coefficient   Cancel   Lcr = 2 lo   Cancel   Lcr = lo   Help   Lcr = 0.5 lo   Lcr = 1.00   Intermediate bracings	Lateral buckling parameters       More         ✓ Lateral buckling       Lateral buckling length coefficient         Load level:       Upper flange         Lor = (Lcr1,Lcr2,)       Lcr = lo         ④ General method       [6.3.2.2]       Lambda LT.0 = 0.40         ○ Detailed method       [6.3.2.3]       Beta =       0.75         □ Simplified method for beams with kfl =       1.10       1.10         Additional sets of member parameters       Service       Complex         □ Limit deflections and displacements:       Service       Note         □ Ihin-walled sections:       Thin-walled       Note         □ Fire analysis parameters:       Eire       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.

Member Definition - Additional Paramet	ers 🗙			
Load parameters Load type: MY A MZ	OK Cancel			
Section parameters	Help			
Anet/Agross ratio: 1,00				
Shear parameter Eta: 1,00				
Angles in tension [6.2.3.5]		💋 Load Type		
Connected by one bolt row				01
Number of bolts n: 2		unife	orm or varying linearly moment	OK
Diameter of bolt openings d0: 1,00 cm			orm load -	Cancel
Distances between bolts p1: 3,00 cm			oly supported beam	
Distance between bolt and angle gdge e2: cm			orm load - 1 beam	Help
Pipes <u>H</u> ot-rolled pipes			centrated force in the center -	
Yield strength:			oly supported beam	
			centrated force in the center -	
C <u>A</u> verage f_ya = 235,00 MPa		fixed	d beam	
Additional conditions for round pipes		2 co	incentrated forces in the	
, The second bending				

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.

#### Autodesk Robot Structural Analysis Professional - Verification Manual for EU Codes

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

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 - Member Verification ( ULS ) 1										
	Results   Message:	s							Calc. Note		
	Member	Section	Material	Lay	Laz	Ratio	Case		Help		
	1 Bar 1	UB 762x267x1	S275	35.35	193.55	0.91	1 STA1				
									Analysis Map		
									Calculation points Division: n = 101		
1									Extremes: none		
I									Additional: none		

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

FRESULTS - Code - EN 1993-1:2005	_ 🗆 🗙	📁 RESU	LTS - Code -	EN 1993-1:	2005			_ 🗆 ×
Auto         Bar:         1 Bar 1           UB 752x257x173         V         Order         54 / y= 053 L = 5.72 m           Load case:         1 STA1	ОК		2x267x173	<u>buto</u>	Bar: Point / Coo Load case		•	<u>ОК</u>
Simplified results Detailed results	Change	Simplifi	ed results De	atailed results	1			Change
FORCES My.Ed = 1276 7 NYm My.p.I.Rd = 1276 5 NYm		S	ymbol	Values	Unit Cross-	Symbol description Section	1	
My,c,Rd = 1703.6 kN*m Vz,Ed = -251.5 kN Vz,c.Rd = 1825.9 kN	Eorces	Ax		220.000	cm2	Cross-section area		Eorces
Mb,Rd = 1401.2 kN*m		Ay		121.931	cm2	Shear area - Y-axis		Detailed
Class of section = 1	Detailed	Az		115.002	cm2	Shear area - Z-axis		Detated
LATERAL BUCKLING		bx		267.000	cm4	Torsional constant		
Image: Second state         Z = 1.00         Mcr = 4311.9 kN*m         Curve_LT - b         XLT = 0.82           Image: Second state         Lcrusce=5.10 m         Lam LT = 0.63         fiLT = 0.77		ly	2	05300.000	cm4	Moment of inertia of a section about the Y-axis		
Lcr.upp=5.10 m Lam_LT = 0.63 fiLT = 0.77		lz		6850.000	cm4	Moment of inertia of a section about the Z-axis		
BUCKLING Z	Calc. Note	Wphy	(	6195.000	cm3	Plastic section modulus about the Y (major) axis		Calc. Note
$\mathbf{X}$	<u>Calc. Note</u>	Wpla	2	807.000	cm3	Plastic section modulus about the Z (minor) axis		<u>calc. Note</u>
	Parameters	h		76.20	cm	Height of cross-section		Parameters
		b		26.67	cm	Width of cross-section		
	Help	tf		2.16	cm	Flange thickness		Help
SECTION CHECK	nep	tw		1.43	cm	Web thickness		nep
MuEd/MuEd = 0.75 < 1.00 (6.2.5.(1))		ry		30.55	cm	Radius of gyration - Y-axis		
Vz,Ed/Vz,c,Rd = 0.14 < 1.00 (6.2.6.(1))		rz		5.58	cm	Radius of gyration - Z-axis		
MEMBER STABILITY CHECK						Material:		
My,Ed/Mb,Rd = 0.91 < 1.00 (6.3.2.1.(1))		Nam	e			\$275 (\$275)		
		ty	1	275.00	MPa	Design yield strength of material (3.2)	-	
	-						_	1

## STEEL DESIGN

CODE: EN 1993-1:2005, Eurocode 3: Design of steel structures. ANALYSIS TYPE: Member Verification										
CODE GROUP: MEMBER: 1 Bar 1 5.72 m	POINT: 54	COC	<b>DRDINATE:</b> $x = 0.53 L =$							
LOADS: Governing Load Case: 1	STA1									
<b>MATERIAL:</b> S275 (S275) fy = 275	5.00 MPa									
<b>SECTION PARA</b> h=76.20 cm b=26.67 cm tw=1.43 cm tf=2.16 cm	AMETERS: UB 762x267x1 gM0=1.00 Ay=121.931 cm2 Iy=205300.000 cm4 Wply=6195.000 cm3	<b>73</b> 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 AI	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							
z = 1.00	BUCKLING PARAMETER Mcr = 4311.9 kN*m Lam_LT = 0.63	<b>S:</b> Curve,LT - b fi,LT = 0.77	XLT = 0.82							
BUCKLING PARAMET	ERS:	About Z axis:								
VERIFICATION FORMU Section strength check: My,Ed/My,c,Rd = 0.75 < 1 Vz,Ed/Vz,c,Rd = 0.14 < 1. Global stability check of n My,Ed/Mb,Rd = 0.91 < 1.0	1.00 (6.2.5.(1)) 00 (6.2.6.(1)) <i>nember:</i>									

#### Section OK !!!

### COMPARISON: Critical segment CD

Resistance, interaction expression	Robot	Handbook
1. Critical moment for lateral-torsional buckling Mcr	4311,9	4311
2. Reduction factor for lateral-torsional buckling X <sub>LT</sub>	0,82	0,82

## VERIFICATION EXAMPLE 5 - Combined bi-axial bending and compression

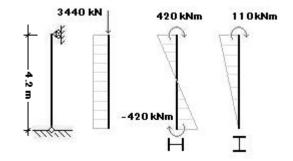
Example taken from Designer's Guide to EN 1993-1-1 L.Gardner and D.A.Nethercot, Thomas Telford Publishing, 2005

#### 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*.

🗾 Definitions - EN 1993-1:2005	<b>5</b> Buckling Diagrams	×
Members       Groups         Number:       1         Basic data         Bar list:         I         Name:         Bar 1         C. Group:         Member type:         Column1         OK         Save         Help	$ \begin{array}{c ccccccccccccccccccccccccccccccccccc$	OK 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 1993-1:2	2005	×
Member <u>type:</u> Column1			Save
Buckling (Yaxis) Member length ly:	Buckling (Zaxis) Memberler	igth lz:	Close
C <u>B</u> eal ⊙ <u>C</u> oefficient 1,00	C Re <u>a</u> l ⊙ C <u>o</u> efficient [1]	00	
Buckling length coeff. Y:	Buckling length	i coeff. Z:	
Non-sway Buckling curve Y auto	N Bucking <u>c</u> urve Z	on-sway auto 💌	
Elexural-torsional buckling of m	nonosymmetric sectio	ins	
Lateral buckling parameters           Image: Lateral buckling           Load level:	Lateral buckling leng	gth coefficient ower flange	More
• <u>G</u> eneral method [6.3.2.2]	Lambda 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]	ith k <u>f</u> l =	1,10	
Additional sets of member paramet	ers		
Limit deflections and displacer	nents:	<u>d</u> ervice	
Comple <u>x</u> sections:	<u>[</u>	Complex	
<u>I</u> hin-walled sections:	Tł	in-walled	Note
<u>F</u> ire 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
C Code group verification:	List
C Code group <u>d</u> esign:	List
Optimization	Options
Loads C <u>a</u> ses: 1	List Limit state
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.

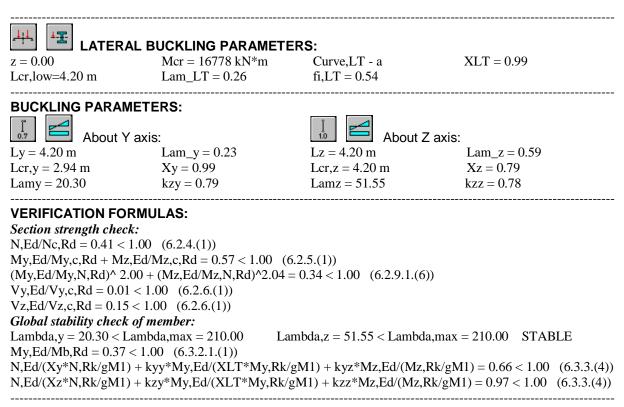
<b>5</b> EN 1993-	l:2005 -		_ 🗆 🗵							
Results M	essages	C <u>a</u> lc. Note	[]							
Memb	er	Section	Material	Lay	Laz	Ratio	Case			Help
1 Bar	1 🛛	UC 305x305x2	S 275	20.30	51.55	0.97	1 STA1		⊢ Ratio	
									Calculation p Division: Extremes: Additional:	Map n = 3 none none

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.

ESULTS - Code - EN 1993-1:2005	_ 🗆 🗡	ESULTS - Code - EN 1993-1:2005	
Auto         Bar:         1 Bar 1           UC: 305:0250:240         ▼         Section DK         ●           Local case:         1 57A1         ●         ●	<u> </u>	Auto         Section 0K           UC 305x305x240         Y	0K
Simplified results Detailed results	Change	Simplified results Detailed results	Change
FBRICS         V/Ed= 420 INYm         MEEd= 110 INYm         V/Ed= 26 IM           NEA= 3040 N         MyEd= 420 INYm         MEEd= 110 INYm         V/Ed= 26 IM           NEA= 30415 N         MyEd= 1167 INYm         MEEd= 355 IMY         V/Ed= 260 IM           NEA= 30415 N         MyEd= 1167 INYm         MEEd= 355 IMY         V/Ed= 260 IM		Symbol Values Unit Symbol description Section Cross-section properties: UC 305x205x240	
My,N,Rd = 774 kN'm Mz,N,Rd = 503 kN'm Vz,c,Rd = 1372 kN	Eorces	Ax 306.000 cm2 Cross-section area	Eorces
Mb,Rd = 1150 kN*m	Detailed	Ay 249.236 cm2 Shear area - Y-axis	Detailed
Class of section = 1	Detgied	Az 86.435 cm2 Shear area - Z-axis	Detgied
LATERAL BUCKLING		bx 1271.000 cm4 Torsional constant	
z = 0.00         Mcr = 16778 kN*m         Curve.LT - a         XLT = 0.99           Lor low=4 20 m         Low LT = 0.26         61 T = 0.54		ly 64200.000 cm4 Moment of inertia of a section about the Y-axis	
Lcr,Jow=4.20 m Lam_LT = 0.26 fi,LT = 0.54		Iz 20310.000 cm4 Moment of inertia of a section about the Z-axis	
- BUCKLING Y	Calc. Note	Wply 4243.000 cm3 Plastic section modulus about the Y (major) axis	Calc. Note
r Ly = 4.20 m Lam_y = 0.23 r Lz = 4.20 m Lam_z = 0.59	- cgic. Hoto	Wplz 1945.000 cm3 Plastic section modulus about the Z (minor) axis	- Cgic. Hoto
0.7 Lor.y = 2.94 m Xy = 0.99 10 Lor.z = 4.20 m Xz = 0.79	Parameters	h 35.26 cm Height of cross-section	Parameters
Lamy = 20.30 kzy = 0.79 Lamz = 51.55 kzz = 0.78		b 31.79 cm Width of cross-section	
	Help	tf 3.77 cm Flange thickness	Help
SECTION CHECK		tw 2.30 cm Web thickness	
My,Ed/My,c,Rd + Mz,Ed/Mz,c,Rd = 0.57 < 1.00 (6.2.5.(1))		ry 14.48 cm Radius of gyration - Y-axis	
Vz,Ed/Vz,c,Rd = 0.15 < 1.00 (6.2.6.(1))		rz 8.15 cm Radius of gyration - Z-axis	
MEMBER STABILITY CHECK Lamy = 20.30 < Lam,max = 210.00 Lamz = 51.55 < Lam,max = 210.00 STABLE		Material:	
N,Ed/(Xz*N,Rk/gM1) + kzy*My,Ed/(XLT*My,Rk/gM1) + kzz*Mz,Ed/(Mz,Rk/gM1) = 0.97 < 1.00 (6.3.3.(4))		Name \$ 275 (\$275)	
		fy 275.00 MPa Design yield strength of material (3.2)	

### STEEL DESIGN

CODE: EN 1993-1:2005, Eurocode 3: Design of steel structures. ANALYSIS TYPE: Member Verification										
CODE GROUP: MEMBER: 1 Bar 1	POINT: 3	COORD	<b>DINATE:</b> $x = 1.00 L = 4.20 m$							
LOADS: Governing Load Case:	1 STA1									
<b>MATERIAL:</b> S 275 (S275) fy =	275.00 MPa									
	ARAMETERS: UC 305x305x									
h=35.26 cm	gM0=1.00	gM1=1.00								
b=31.79 cm	Ay=249.236 cm2	Az=86.435 cm2	Ax=306.000 cm2							
tw=2.30 cm tf=3.77 cm	Iy=64200.000 cm4 Wply=4243.000 cm3	Iz=20310.000 cm4 Wplz=1945.000 cm3	Ix=1271.000 cm4							
INTERNAL FORCES	AND CAPACITIES:									
N,Ed = 3440 kN	My,Ed = -420  kN*m	Mz,Ed = 110 kN*m	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 = $6640 \text{ kN}$	My,c,Rd = 1167 kN*m	Mz,c,Rd = 535 kN*m	Vz,Ed = -200  kN							
	My,N,Rd = 774 kN*m	Mz,N,Rd = 503 kN*m	Vz,c,Rd = 1372 kN							
	Mb,Rd = 1150 kN*m		Class of section $= 1$							
			$C_{1035}$ Of $SCC10II = 1$							



#### 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

Autodesk Robot Structural Analysis Professional - Verification Manual for EU Codes

## CONCRETE

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

## VERIFICATION EXAMPLE 1 - Dimensioning reinforcement in rectangular section at bending

Example based on:

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

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

#### DESCRIPTION OF THE EXAMPLE:

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

#### **GEOMETRY**:

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

#### MATERIAL:

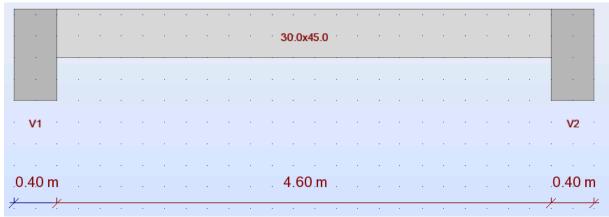
Concrete: C25/30  $\alpha_{cc} = 0.85$ Steel: fyk=355 [MPa]

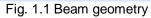
#### LOADS:

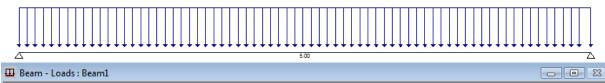
Bending moment M = 100kNm [cm<sup>2</sup>]

#### **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*).







2													
		Case number	Distributed load	Nature	Li st	Positi on	Coord. syste	Load directi	Load factor	Coordinates	Projection	x1 (m)	p1 (kN/m)
	1	DL1	uniform	dead loa	1	upp	Local	Z	1.35	absolute	Not projected		23.70
	÷												

Fig. 1.2 Loads and the calculation model

Tob Preferences				? 💌
	AULTS	Database Name	Database Description	■
Bolts     Anchor bolts     Wire fabrics     The sign codes     The sign codes	EC2 - ITALI NEN-EN PN 2002_#	III		4
👻 <u>O</u> pen default	parameters			
Save current para	meters as default	ОК	Cancel	Help

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

Han Job Preferences		? 🔀		
🚅 🖶 🗙 Ӿ 🛛 🛛	EFAULTS	•		
<ul> <li>Units and Formats</li> <li>Materials</li> <li>Databases</li> <li>Design codes</li> <li>Loads</li> <li>Structure Analysis</li> <li>Modal Analysis</li> <li>Seismic Analysis</li> <li>Work Parameters</li> </ul>	<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 • • • • • • • • • • • • • • • • • •		
Image: Contract of the second seco				
Partial Factors for a Code EN 1				

EN 1992-1-1:2004		Coefficient	Value	Code reference	1
	8	k1 (redistribution)	0.44	1992-1-1 5.5 (4)	
SFS-EN 1992-1-1	9	k2 (redistribution)	Auto	1992-1-1 5.5 (4)	
UNI-EN1992-1-1	10	k3 (redistribution)	0.54	1992-1-1 5.5 (4)	
PN-EN 1992-1-1:2008	11	k4 (redistribution)	Auto	1992-1-1 5.5 (4)	=
FN-EN 1332-1-1.2000	12	k5 (redistribution)	0.70	1992-1-1 5.5 (4)	ľ
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 <sub>ct</sub>	1.00	1992-1-1 3.1.6 (2)P	
	16	$\epsilon_{ud} / \epsilon_{uk}$	0.90	1992-1-1 3.2.7 (2)	-
			Copy to user's set		

Fig. 1.4 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. $A_{sl}$	$8.53 \text{ cm}^2$	$8.57 \text{ cm}^2$

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

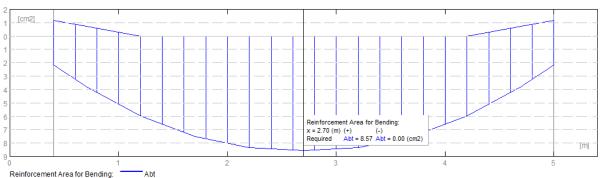


Fig. 1.5. Theoretical (required) areas of reinforcement in beam.

#### 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	$\gamma_{s}$	$\alpha_{cc}$	bottom reinf. $A_{sl}$ -Robot results
Handbook example	1.5	1.15	0.85	$8.57 \text{ cm}^2$
(general Eurocode 2 edition				
with modified $\alpha_{cc}$				
EN 1992-1-1:2004	1.5	1.15	1.0	$8.45 \text{ cm}^2$
AC:2008				
PN-EN 1992-1-1:2008	1.4	1.15	1.0	$8.41 \text{ cm}^2$
UNI-EN 1992-1-1	1.5	1.15	0.85	$8.57 \text{ cm}^2$
SFS-EN 1992-1-1	1.5	1.15	0.85	$8.57 \text{ cm}^2$
EN 1992-1-1 DK NA:2007	1.45	1.2	1.0	$8.80  {\rm cm}^2$
BS EN1992-1-1:2004	1.5	1.15	0.85	$8.57 \text{ cm}^2$
NA2005				
NS-EN 1992-1-	1.5	1.15	0.85	$8.57 \text{ cm}^2$
1:2004/NA:2008				
NF EN 1992-1-1/NA:2007	1.5	1.15	1.0	$8.45 \text{ cm}^2$

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

### **VERIFICATION EXAMPLE 2** - Dimensioning reinforcement in rectangular section at bending

Example based on:

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

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

#### **DESCRIPTION OF THE EXAMPLE:**

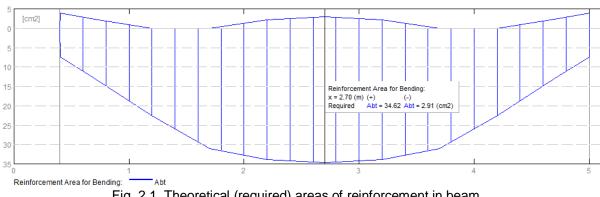
Calculate the reinforcement in rectangular section at simple bending at ULS. In this example, the results of the program are compared against [1]. The data is the same as in Verification problem 1, except for the bending moment which is equal to M=320 kNm.

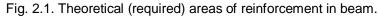
#### **RESULTS OF LONGITUDINAL REINFORCEMENT (REINFORCEMENT FOR BENDING)** CALCULATION:

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

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

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





#### ANALYSIS OF RESULTS FOR NADs:

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

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

Code	24	~	a	bottom reinf. $A_{sl}$ -	top reinf. $A_{s2}$ -
Code	γc	$\gamma_{\rm s}$	$\alpha_{cc}$	Robot results	Robot results
Handbook example	1.5	1.15	0.85	$34.73 \text{ cm}^2$	$2.92 \text{ cm}^2$
(general Eurocode 2	1.5	1.15	0.85	54.75 CIII	2.92 CIII
edition with					
modified $\alpha_{cc}$				2	
EN 1992-1-1:2004	1.5	1.15	1.0	$34.27 \text{ cm}^2$	$0.0 \text{ cm}^2$
AC:2008					
PN-EN 1992-1-	1.4	1.15	1.0	$33.09 \text{ cm}^2$	$0.0 \text{ cm}^2$
1:2008					
UNI-EN 1992-1-1	1.5	1.15	0.85	$34.73 \text{ cm}^2$	$2.92 \text{ cm}^2$
SFS-EN 1992-1-1	1.5	1.15	0.85	$34.73 \text{ cm}^2$	$2.92 \text{ cm}^2$
EN 1992-1-1 DK	1.45	1.2	1.0	$35.12 \text{ cm}^2$	$0.0 \text{ cm}^2$
NA:2007					
BS EN1992-1-	1.5	1.15	0.85	$34.73 \text{ cm}^2$	$2.92 \text{ cm}^2$
1:2004 NA2005					
NS-EN 1992-1-	1.5	1.15	0.85	$34.73 \text{ cm}^2$	$2.92 \text{ cm}^2$
1:2004/NA:2008					
NF EN 1992-1-	1.5	1.15	1.0	$34.27 \text{ cm}^2$	$0.0 \text{ cm}^2$
1/NA:2007					

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

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

Example based on:

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

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

#### **DESCRIPTION OF THE EXAMPLE:**

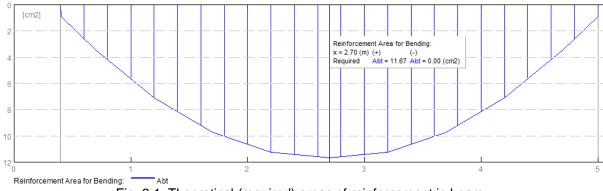
Calculate the reinforcement in rectangular section at bending with compression at ULS. In this example, the results of the program are compared against [1]. The data is the same as in Verification problem 1, except of the forces which are: bending moment M=150 kNm, and compressive force N=150 kNm.

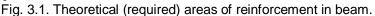
#### **RESULTS OF LONGITUDINAL REINFORCEMENT (REINFORCEMENT FOR BENDING)** CALCULATION:

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

Theoretical areas	[1]	Robot
bottom reinf. $A_{sI}$	$11.62 \text{ cm}^2$	$11.67 \text{ cm}^2$

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





#### ANALYSIS OF RESULTS FOR NAD's:

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

Code	γc	$\gamma_{\rm s}$	$\alpha_{cc}$	bottom reinf. A <sub>s1</sub> -Robot results
Handbook example	1.5	1.15	0.85	$11.67 \text{ cm}^2$
(general Eurocode 2 edition				
with modified $\alpha_{cc}$				
EN 1992-1-1:2004	1.5	1.15	1.0	$11.17 \text{ cm}^2$
AC:2008				
PN-EN 1992-1-1:2008	1.4	1.15	1.0	$11.00 \text{ cm}^2$
UNI-EN 1992-1-1	1.5	1.15	0.85	$11.67 \text{ cm}^2$
SFS-EN 1992-1-1	1.5	1.15	0.85	$11.67 \text{ cm}^2$
EN 1992-1-1 DK NA:2007	1.45	1.2	1.0	$11.57 \text{ cm}^2$
BS EN1992-1-1:2004	1.5	1.15	0.85	$11.67 \text{ cm}^2$
NA2005				
NS-EN 1992-1-	1.5	1.15	0.85	$11.67 \text{ cm}^2$
1:2004/NA:2008				
NF EN 1992-1-1/NA:2007	1.5	1.15	1.0	$11.17 \text{ cm}^2$

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). Dolnoślaskie Wydawnictwa Edukacyjne, Wroclaw 2006, Example 6.10, pp. 334 \*

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

#### **DESCRIPTION OF THE EXAMPLE:**

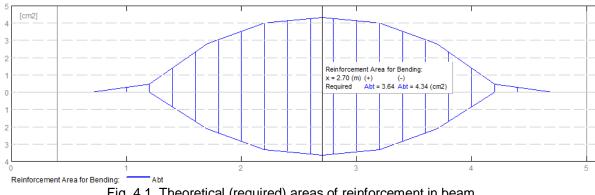
Calculate the reinforcement in rectangular section at bending with compression at ULS. In this example, the results of the program are compared against [1]. The data is the same as in Verification problem 1, except of the forces which are: bending moment M=150 kNm, and compressive force N=1000 kNm.

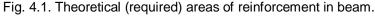
#### RESULTS OF LONGITUDINAL REINFORCEMENT (REINFORCEMENT FOR BENDING) CALCULATION:

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

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

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				bottom painf A	top point A
Code	$\gamma_{\rm c}$	$\gamma_{\rm s}$	$\alpha_{cc}$	bottom reinf. $A_{sl}$ -	top reinf. $A_{s2}$ -
				Robot results	Robot results
Handbook example	1.5	1.15	0.85	$3.64 \text{ cm}^2$	$4.34 \text{ cm}^2$
(general Eurocode 2					
edition with modified					
$lpha_{ t cc)}$					
EN 1992-1-1:2004	1.5	1.15	1.0	$4.75 \text{ cm}^2$	$0.0 \text{ cm}^2$
AC:2008					
PN-EN 1992-1-1:2008	1.4	1.15	1.0	$3.24 \text{ cm}^2$	$0.0 \mathrm{cm}^2$
UNI-EN 1992-1-1	1.5	1.15	0.85	$3.64 \text{ cm}^2$	$4.34 \text{ cm}^2$
SFS-EN 1992-1-1	1.5	1.15	0.85	$3.64 \text{ cm}^2$	$4.34 \text{ cm}^2$
EN 1992-1-1 DK	1.45	1.2	1.0	$4.13 \text{ cm}^2$	$0.0 \mathrm{cm}^2$
NA:2007					
BS EN1992-1-1:2004	1.5	1.15	0.85	$3.64 \text{ cm}^2$	$4.34 \text{ cm}^2$
NA2005					
NS-EN 1992-1-	1.5	1.15	0.85	$3.64 \text{ cm}^2$	$4.34 \text{ cm}^2$
1:2004/NA:2008					
NF EN 1992-1-	1.5	1.15	1.0	$4.75 \text{ cm}^2$	$0.0 \mathrm{cm}^2$
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:

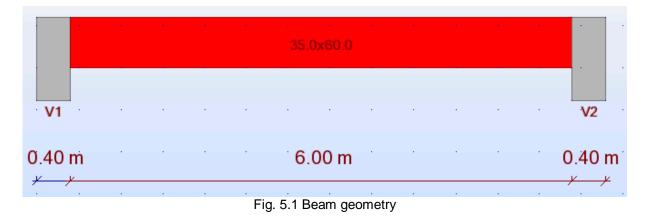
Concrete:	C20/255
Steel:	B500C (f <sub>yk</sub> = 500 [MPa])

#### 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.



# 

_	$\Delta$							6.40					
🗳 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
	ż												

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)   
$$\rho_l &= \frac{A_{sl}}{b_w d} = 0.0117 \\ A_{sl} &= 2199 m m^2 \\ b_w &= 350 m m \\ f_{ck} &= 20 M P a \\ \sigma_{cp} &= 0 M P a \\ But should not be smaller than: \\ V_{Rd,c} &= \left[ V_{\min} + k_1 \sigma_{cp} \right] b_w d = 59.9 k N \end{split}$$

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

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

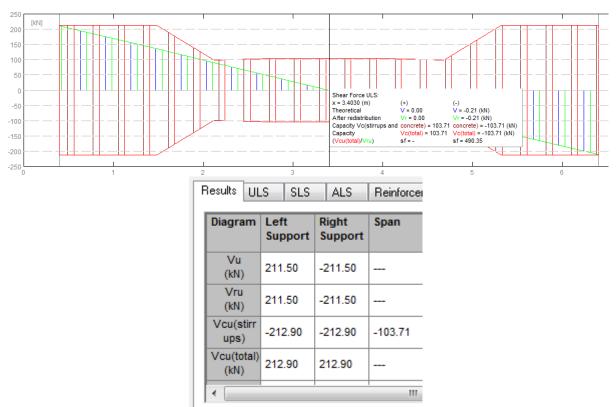


Fig. 5.3 Shear force distribution and shear capacity

#### • 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]:

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

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

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

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

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

$$z = 0.9d = 0.49m$$
(2 bars \phi 8)

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

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

$$f_{ywk} = 500MPa$$

$$\gamma_s = 1.15$$

$$\cot \theta = 1.0$$

(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:

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

And should not be greater than: VRd, max =  $\frac{\alpha_{cw}b_w z v_1 f_{cd}}{\cot \theta + \tan \theta} = 627.4kN$ 

$$\alpha_{cw} = 1.0$$

$$v_1 = 0.552$$

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

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

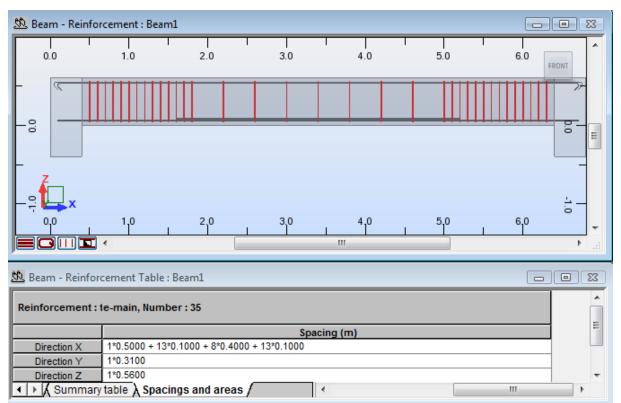


Fig. 5.4 Shear reinforcement distribution (see Direction X in the Reinforcement table)

#### 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	$\gamma_{s}$	$\alpha_{cc}$	Shear capacity $V_{Rd,c}$	Shear capacity
					$V_{Rd,s}$
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  $\gamma_c$  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  $\gamma_c$  .

\* 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 Professional are based on the last edition with the corrections EN1992-1-1:2004 AC:2008. However, the last correction does not introduce any changes within the range of the calculations presented in the examples.

#### **DESCRIPTION OF THE EXAMPLE:**

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

#### **GEOMETRY:**

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

#### MATERIAL:

Concrete: C16/20

#### **REINFORCEMENT:**

Bottom bars: 5\u00e920

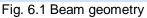
#### LOADS:

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

#### **IMPORTANT STEPS:**

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

			;	30.0x50.0	)			
V1								V2
0.60 m				7.00 m	i			0.60 m



Π		$\downarrow \downarrow \downarrow$		ļ ļ ļ								$\prod_{i=1}^{n}$
Δ	<u>∧</u> 7.50 <u>∧</u>											
	Case umber	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*).

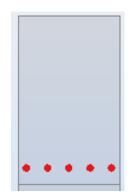


Fig. 5.2 Reinforcement ( $5\phi 20$ ) assumed in [1]

#### **RESULTS OF DEFLECTION CALCULATION:**

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

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

$$\xi = 0.9686$$
  

$$f_{I} = 2.720cm$$
  

$$f_{II} = 3.791cm$$

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

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

#### ANALYSIS OF RESULTS FOR NADs:

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

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

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

# LITERATURE

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

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

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

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

#### Example based on:

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

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

#### **DESCRIPTION OF THE EXAMPLE:**

The example illustrates the influence of second order-effects on the total moment of column AB of the frame (*Fig.1.1*). In [1], the reinforcement is assumed *a priori*. We 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.

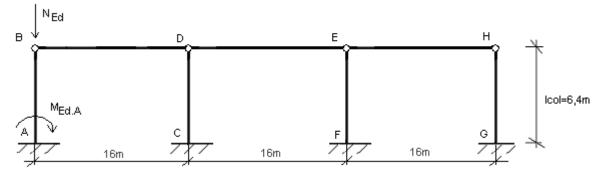


Fig. 1.1. The model of the frame with the analyzed column AB.

#### **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:

#### LOADS:

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

#### **REINFORCEMENT:**

5 bars  $\phi$ 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).

III Buckling Len	gth
Nu	Total structure height: 6.40 m mber of vertical elements m: 4
📄 📄 Direction Y =	
🗖 Off	
	Structure Non-sway Sway $L_y = 6.40$ m $k_y = 2.00$

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)*.

🖫 Job Preferences				? 💌
🗃 层 🗙 🔆 🛛 DEFAL	ILTS			•
- Steel and timber s	Database	Database Name	Database Description	
Vehicle loads Standard loads Building soils Building soils Anchor bolts Reinforcing bars Wire fabrics Design codes	EN 1992-1-1 EC2 - ICEL EC2 - ITALI NEN-EN PN 2002_#			4
🚔 <u>O</u> pen default par	rameters			
Save current paramet	ters 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*).

🔽 De	siq	n for s	simple	ber	ndino	Į.			
۲	M	) dire	ction				O Ma	z directi	on
1 0 0	•			•					

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).* 

Han Job Preferences			[	? 💌
😅 层 🗙 Ӿ 🛛 🖻	FAULTS			-
<ul> <li>Units and Formats</li> <li>Materials</li> <li>Databases</li> <li>Design codes</li> <li>Loads</li> <li>Structure Analysis</li> <li>Modal Analysis</li> <li>Non-linear Analysis</li> <li>Seismic Analysis</li> <li>Work Parameters</li> </ul>	<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:	ENV 1997-1	8:2005 • I-1:1992 • 1:2004 AC:2008 •	
🔫 <u>O</u> pen defaul	lt parameters			
尾 <u>S</u> ave current para	ameters as default	ОК	Cancel H	elp
R <sup>10</sup> Partial Factors for a Code EN 19 RC structures	92-1-1:2004 AC:2008			? 💌
EN 1992-1-1:2004	Coefficient	Value	Code reference	•

EN 1992-1-1:2004		Coefficient	Value	Code reference	*
	8	k1 (redistribution)	0.44	1992-1-1 5.5 (4)	
SFS-EN 1992-1-1	9	k2 (redistribution)	Auto	1992-1-1 5.5 (4)	
UNI-EN1992-1-1	10	k3 (redistribution)	0.54	1992-1-1 5.5 (4)	
PN-EN 1992-1-1:2008	11	k4 (redistribution)	Auto	1992-1-1 5.5 (4)	Ξ
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 <sub>ct</sub>	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
		Сору	to user's set		
			OK	Cancel He	lp )

Fig. 1.8 Definition of partial factors

NOTE: The program automatically generates smaller reinforcement (8  $\phi$ 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*)

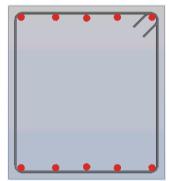


Fig. 1.9 Reinforcement assumed for the calculation (10  $\phi$ 20).

	(Unit)	[1]	Robot (results presented in calculation note)
$\lambda_{ m lim}$	(-)	32.2	32.3
$\alpha_{_h}$	(-)	0.791	0.791
$\alpha_{m}$	(-)	0.791	0.791
e <sub>a</sub>	(cm)	2.0	2.0
K <sub>r</sub>	(-)	1.0	1.0
$K_{\varphi}$	(-)	1.0	1.0
$1/r_0$	(1/m)	0.00863	0.00853*
$1/r_0$	(1/m)	0.00863	0.00853
с	(-)	10	10
$e_2$	(cm)	13.7 (14.1)**	14.0
$M_{Ed}$	(kNm)	289.8 (293.9)**	291.97

#### **RESULTS OF BUCKLING ANALYSIS - NOMINAL CURVATURE METHOD:**

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

NOTES ON DIFFERENCES IN THE COMPARISON:

\* - the difference is due to accuracy of steel strength value used in calculation of  $1/r_0$  (the

authors of [1] use fixed  $f_{yd} = 350 MPa$  value, while program uses  $f_{yd} = f_{yk} / \gamma_s = 357 MPa$ 

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

#### ANALYSIS OF RESULTS FOR NADs:

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

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

#### **RESULTS OF BUCKLING ANALYSIS - NOMINAL STIFFNESS METHOD:**

	(Unit)	[1]	Robot (results presented in calculation note)
$J_s$	(cm <sup>4</sup> )	14500	14442
$J_{c}$	(cm <sup>4</sup> )	785000**	468750
EJ	(kNm <sup>2</sup> )	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,  $\beta$  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	γc	$\alpha_{cc}$	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 Professional are based on the last edition with the corrections EN1992-1-1:2004 AC:2008. However, the last correction does not introduce any changes within the range of the calculations presented in the examples.

#### **DESCRIPTION OF THE EXAMPLE:**

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

#### **GEOMETRY:**

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

#### **REINFORCEMENT:**

reinforcement area:	$A_x = A_y = 16.08$	[cm²/m]
reinforcement ratio:	ρ <sub>x</sub> =ρ <sub>y</sub> =0.0077	

#### MATERIAL:

Concrete:	fck = 15	[MPa]
Concrete.	10K = 10	נועוד מ

#### **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	eous Orthotropic
	h
1	
Label:	h240 Color: Auto 🗸
© Co	onstant Th = 24.0 (cm)
© Va	ariable along a line
🔘 Va	ariable on a plane
	Point coordinates Thicknesses
P1:	(m) (cm) 7.0000; 2.0000; 0.0000 0.0
P2:	7.0000; 2.0000; 0.0000 0.0
P3:	7.0000; 2.0000; 0.0000 0.0
	eduction of the 1.00 ×1g >>
m m	oment of inertia
	Parameters of foundation elasticity
<u>M</u> aterial:	C20/25
	Add Close Help

Fig. 1.1 Slab thicknes

Reg Material Definition	A Dicese	? ×
Steel Concrete Aluminum Timt	ber Other	
<u>N</u> ame: fck15 ·	▼ Descripti <u>o</u> n: EC2 Concrete C12/15	
Elasticity Young modulus, <u>E</u> :	27000.00 (MPa)	
Poisson ratio, <u>v</u> :	0.2 Characteristic • 15.00	MPa)
Shear modulus, <u>G</u> :	11250.00 (MPa) Sample: Cylindrical	-
Force density (unit weight):	24.53 (kN/m3)	
Ihermal expansion coefficient:	0.000010 (1/°C)	
Damping ratio:	0.04	
Add	Delete OK Cancel	Help

Fig. 1.2 Definition of new concrete type

F EN 1992-1-1:2	2004 AC:2008 Rein	forceme	nt Par 🖢	<u> </u>
General Materia	als SLS Parameters	s Reinfo	rcement	
		<u>(a1</u>		<u>d2'</u> ₹_ <u>d2</u>
-Bar dimensions				
d1:	12 -	d2:	12	•
d1':	12 🔻	d2':	12	•
Cover (cm)				
c1:	1.9	c2:	1.9	
c1':	0.0	c2 ':	0.0	
	Deviat	ions		
Unidirectional	reinforcement			
Membrane reir	nforcement in one lay	/er		
- Minimum reinfor	cement			
🔘 None				
Sor FE for which reinforcement As>0				
For the whole	) panel			
No	ote Add		Close	Help

Fig. 1.3 Definition of covers of reinforcement

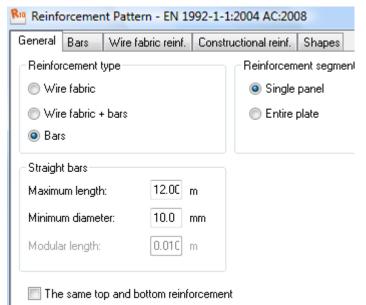


Fig. 1.4 Selection of reinforcement with bars

Reinforcement Pattern - EN 1992-1-1:2004 AC:2008							
General Bars	Wire fabric reinf.	Con	structior	al reinf.	Shapes		
- Bottom reinforce	ement			Top rein	forcement	t	
Diameter				Diame	ter		
Direction X	12		>>	Direct	ion X	12	-
Direction Y	12 •	•		Direct	ion Y	12	•
-Spacing (cm)-		5		Spacir	ng (cm)		
Direction >	< 10.0		<<	🔽 Dir	ection X	7.0	
Direction 1	Y [10.0			🔽 Dir	ection Y	7.0	
Preferred reinfor	cement spacing						
Direction X—							
🔲 Maximum	40.0	cm		Minimun	1	3.0	cm
Direction Y							
🔲 Maximum	45.0	cm		Minimun	ı	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.

न Plat	e and Shell R	einforcer	nent			
Punching						
Veri	ication points			Point grouping	]	
Nar	ne: S1	30x30		List:	-	
			>>	New	Delete	
	New	Delete		Maximum punching	force (kN)	
Тур	e: inter	mal	•	160.00		
Pos	Position (m)					
x =	2.0000 y	= 2.000	0	Node number:	2	
Pur	nching:	from t	oottom	T		
	Head			Dimension	s (cm)	
l r	Туре		b 7 - 9		0.0	
	rectangula	16	$\sim$	<b>h</b> b= 3	:0.0	
	circular h = 0.0					
	Qadm	Q	u	Reinforcement		
	(kN)	(kN)	u (m)	(m), (cm2) / n x (	L Oadm / O L	
S	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	γc	Punching capacity
EN 1992-1-1:2004	1.5	430 kN
AC:2008		
PN-EN 1992-1-1:2008	1.5	430 kN
UNI-EN 1992-1-1	1.4	460 kN
EN 1992-1-1 DK NA:2007	1.45	445 kN
BS EN1992-1-1:2004	1.5	430 kN
NA2005		
NS-EN 1992-1-	1.5	430 kN
1:2004/NA:2008		
NF EN 1992-1-1/NA:2007	1.5	457 kN

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

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

Example based on: Manual calculation

#### DESCRIPTION OF THE EXAMPLE:

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

#### **GEOMETRY:**

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

#### **REINFORCEMENT:**

reinforcement area:	$A_x = A_y = 16.08$	[cm <sup>2</sup> ]
reinforcement ratio:	ρ <sub>x</sub> =ρ <sub>y</sub> =0.0077	

#### MATERIAL:

Concrete: C20/25

#### FORCES IN THE NODE:

Vertical force:	N = 192 kN
Moments:	$M_x = 24 \text{ kN}$
	$M_y = 40 \text{ kN}$

#### 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/m<sup>3</sup>)

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

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

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

$$A = 0.426m^2$$

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

		Manual Ilculation	Ro	obot	
Punching capacity		211 kN	21	1 kN	
Plate and Shell Reinforcem	ient			<u> </u>	
Punching					
Verification points		- Point groupin	g		
Name: S1 30x30	$\rightarrow$	List:		•	
	>>	New		Delete	
New Delete		Maximum pu	inching force	e (kN)	
- Position (m)		0.0	0		
	Position (m)           x =         0.0000         y =         0.0000				
Punching: from to	P	Ŧ			
- Head		– Dim	ensions (cm	J	
Туре	ه		a = 0.0		
rectangular	*	T/h	<b>b</b> = 0.0	$\dashv$	
🔘 circular	H			=	
			n = 0.0		
Qadm Q (kN) (kN)	u (m)	Reinforce (m), (cm2		Qadm / Q	
<b>S1</b> 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 <sub>x</sub> =50	[cm]
	c <sub>y</sub> =30	[cm]

#### **REINFORCEMENT:**

reinforcement area:	$A_x = A_y = 16.08$	[cm <sup>2</sup> ]
reinforcement ratio:	ρ <sub>x</sub> =ρ <sub>y</sub> =0.0077	

#### MATERIAL:

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

#### FORCES IN THE NODE:

Vertical reaction:	V = 192 kN
Moments:	$M_x = 24 \text{ kN}$
	$M_y = 40 \text{ kN}$

#### CALCULATION OF $\beta$ COEFFICIENT:

In Robot,  $\beta$  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).

			Manual calculation	F	Robot			
	Punching force			342 kN	34	45 kN		
	Plate ar	nd Shell F	leinforcer	ment			_ 0 ×	3
	<sup>D</sup> unching	]						
	-Verifical	tion points			- Point grouping			
	Name:	S1 !	50x30		List:		-	
				>>	New		Delete	
	Ne	w	Delete		Maximum pun	ching force	e (kN)	
	Type:	unk	nown	-	0.00			
	- Position		0.000		No do un un bor			
	x = 0.1		= 0.000		Node number	r:		
	Punching: from top				<b>T</b>			
	- Hea	ıd			⊂ Dime	nsions (cm	J	
	Type			b - 1-0	a			
	rectangular		H	Ин ь	- 0.0			
	circular			h	- 0.0			
		Qadm (kN)	Q (kN)	u (m)	Reinforcen (m), (cm2)/		Qadm / Q	
	S1	436.47	344.87	4.2264			1.27 > 1	
							,	
	)							
					0	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 Professional are based on the last edition with the corrections EN1992-1-1:2004 AC:2008. However, the last correction does not introduce any changes within the range of the calculations presented in the examples.

#### **DESCRIPTION OF THE EXAMPLE:**

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

#### GEOMETRY:

slab thickness:	h=24.0 [cm]
spacing of columns:	l <sub>x</sub> = 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 <sub>ck</sub> = 20 [MPa]
Steel:	f <sub>yk</sub> = 355 [MPa] (18G2 steel)

#### LOADS:

dead loads:	7.5 kN/m <sup>2</sup>
live loads:	3.0 kN/m <sup>2</sup>
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  $\beta$  coefficient. Instead, the simplified rule (Fig. 6.21N from Eurocede 2 [2]) is used and  $\beta$  =1.15 is assumed. Robot calculations of punching stress are based on calculation of  $\beta$  from equation (6.39), [2]. Thus, in the presented example, the loads as defined cause no bending moments at the support, hence  $\beta$  =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  $\beta$  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			х
Homogene	ous Orthotropic	]		
·	on one pe			
		_		
1			_/	
•				
Label:	h240	Color:	Auto 👻	
💿 Co	nstant	Th = 24.	0 (cm)	
🔘 Va	riable along a line	:		
🔘 Va	riable on a plane			
	Point coo (m		Thicknesses (cm)	
P1:	0.0000; 0.0000	•	0.0	
P2:	0.0000; 0.0000	); 0.0000	0.0	
P3:	0.0000; 0.0000	); 0.0000	0.0	
	eduction of the	1.00	×lg >>	
mo	oment of inertia			
		() L.C.	alaatiaitu	
	Decembers			
	Parameters c		-	
Material:	Parameters o	C20/	-	]

Fig. 4.1. Slab thickness dialog

🖫 Job Preferences				? 💌
🖙 🖶 🗙 🔆 DEFAL	JLTS			•
<ul> <li>➡ Units and Formats</li> <li>▲</li> <li>▲</li> <li>Materials</li> <li>■ Databases</li> </ul>		2		
- Steel and timber si	Database	Database Name	Database Description	
<ul> <li>Vehicle loads</li> <li>Standard loads</li> <li>Building soils</li> <li>Bolts</li> <li>Anchor bolts</li> <li>Reinforcing bars</li> <li>Wire fabrics</li> <li>Design codes</li> </ul>	EN 1992-1-1 EC2 - ICEL EC2 - ITALI NEN-EN PN 2002_#			4
🚔 <u>O</u> pen default pa	rameters			
Save current paramet	ters as default	ОК	Cancel	Help

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

<mark>F</mark> EN 1992-1-1	:2004 AC:2008 R	einforcement	t Para 🗖 🗉 🎫
General Mate	erials SLS Param	eters Reinfo	rcement
	**************************************		
-Bar dimensior	ns		
d1:	12 -	d2:	12 -
d1':	12	d2':	12 -
Cover (cm)			
c1:	1.8	c2:	1.8
c1':	0.0	c2 ':	0.0
	De	eviations	
Unidirection	al reinforcement		
Membrane r	einforcement in on	ie layer	
- Minimum reinl	forcement		
🔘 None			
For FE for	which reinforceme	nt As>0	
For the wh	ole panel		
	Note A	dd (	Close Help

Fig. 4.3 Definition of covers of reinforcement

Reinforcement	Pattern - EN 1992-1	1:2004	AC:2008		
General Bars V	Wire fabric reinf. Con	structio	nal reinf. Shapes		
- Bottom reinforcem	ent		-Top reinforcemen		ОК
Diameter			Diameter		Cancel
Direction X	Auto 👻	>>	Direction X	12 🔻	
Direction Y	Auto 👻		Direction Y	12 🗸	Help
Spacing (cm)	10.0	<<	Spacing (cm) Direction X Direction Y	6.0 6.0	▼ Save As
Preferred reinforce					Delete
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 cm<sup>2</sup>, while in Robot it is 4.14 cm<sup>2</sup> (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  $s_r$ =0.75*d*, while in [1], the assumed value is smaller than this maximum.

	[1]	Robot
Punching reinforcement	2 perimeters A=3.96 cm <sup>2</sup>	2 perimeters A=4.14 cm <sup>2</sup>

📻 Plate ar	nd Shell R	einforce	ment	l			
Punching							
Verifical	tion points			Point grouping			
Name:	S1 4	40x40 > List:			-		
			Delete				
Ne	New Delete Maximum punching force (kN)						
Туре:							
	Position (m)           x = 0.0000         y = 0.0000           Node number:						
Punchi	Punching: from top -						
Head Type rectangular circular d d d d d d d d					n)		
	Qadm (kN)	Q (kN)	u (m)	Reinforcement (m), (cm2) / n x ø	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 (Constraint) - Canadian Constraints (Constraint) - Canadian Constraints (Constraint) - Coolbar & Menu - Printout Parameters - Advanced	ARD Begional settings: Working language: Printout language:	Eurocode    English    English	- *
Update Preferences on exit		Accept Cancel	Help

### **B. Job Preferences**

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

Job Preferences			? <mark>x</mark>
EUF	30 Steel/Aluminum structures: Steel connections: Timber structures: <u>B</u> C structures: <u>G</u> eotechnical:	EN 1993-1:2005 EN 1993-1:2005 PN-8-03150 PN-8-03264 (2002) PN-81/8-03020	• III • III • III • III
		More codes	
🙀 <u>O</u> pen default	parameters		
Save current para	meters 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:				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	-			
٠		•		< III	

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

Regulation of Code List				_	x
Codes:				Current codes:	
Steel / aluminum		•	]	Set as current	
Steel / aluminum Steel connections RC Timber Geotechnical Load combinations Snow/wind loads Seismic loads			2	Code BS-EN 1993-1:2005/NA:2008/AC: DS/EN 1993-1:2005/DK NA:2007/ EN 1993-1:2005/AC:2009 ENV 1993-1:1992 NAD Belgium	•
ASD:1989 Ed.9th Add80 BS 5950:2000 BS-EN 1993-1:2005/NA:2008/AC:2009 BS5950	USA France UK UK EC3 UK		٤	ENV 1993-1:1992 NAD Germany ENV 1993-1:1992 NAD Netherland NF EN 1993-1:2005/NA:2007/AC.; NS-EN 1993-1:2005/NA:2008/AC.; PN-EN 1993-1:2006/AC:2009	-
OK Cancel				<u>H</u> elp	

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

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

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

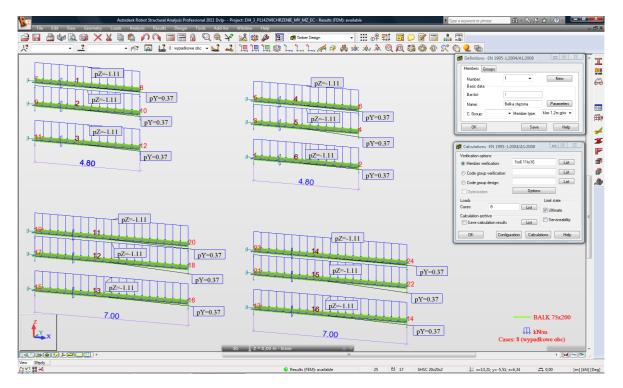
Example based on "Practical design of timber structures to Eurocode 5" Hans Larsen and Vahik Enjily File: EX\_4\_3p114\_bending\_My\_Mz.rtd

### 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.

Definitions -EN 19	95-1:2004/A1:2008	
Members Groups		
N <u>u</u> mber: Basic data	11 💌	New
<u>B</u> ar list:	11	
<u>N</u> ame:	klasyczna stęzona	Parameters
C. <u>G</u> roup:	✓ Member type:	klas 1,2m gór
ОК	<u>S</u> ave	Timber Member Timber Column Timber Beam
		Pret drewniany

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

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

Member Definition - Paramet	ers - EN 1995-1:2004/A1:2008	×		
Member type: Timber Beam		Save		
Buckling (Yaxis) Member length ly:	Buckling (Zaxis) Member length lz:	Close		
<ul> <li><u>Real</u></li> <li><u>Coefficient</u></li> </ul>	<ul> <li>Real</li> <li>Coefficient</li> </ul>	<u>S</u> ervice		
Buckling length coefficient Y:	Buckling length coefficient Z:	<u>M</u> ore		
1,00	1,00	<u>O</u> ther		
Lateral buckling parameters		<u>F</u> ire		
Lat. buckling type:	Lateral buckling length coeff. Upper flange Low <u>e</u> r flange			
Load level:	ld = lo ld = lo			
	eated as solid			
Method of critical stress determina	ation - 6.3.3 :	Note		
Qlassic - formula [6.31]				
Eor rectangular sections - formula [6.32]				
S <u>e</u> rvice class:	1 •	Help		

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

switch on the appropriate Lateral buckling type icon;

💋 Lateral B	X	
IIX	Beam with pinned supports Beam with fixed supports Cantilever Without lateral buckling	OK Cancel Help

select the appropriate Load level icon



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

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

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

💋 Load 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:

🗊 Lateral Buckling Length Coefficient				
	ld = 2lo	OK Cancel		
	ld = lo	Help		
<u> </u>	ld = 0.5 lo			
	<u>i</u> d = 1,00 lo			
X				
Ŧ	Intermediate bracings			

The second one opens BUCKING TYPE DIAGRAMS dialog:

Buckling Type Diagrams	×
$\begin{bmatrix} 1\\10\\0.5\\0.7\\0.9\\\hline \\ \hline \\ 0.9\\\hline \\ \hline \\ \hline \\ 0.9\\\hline \\ \hline $	OK Cancel
<u>C</u> omplex member Batten type: packs	Help
Connection: glued 👻	
Batten coordinates: * L real  relative	

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

🗲 Internal bracings	X
	ling-upper flange
Buckling Y Buckling Z Lateral buckling-upper flange Lateral buckli	ng-lower flange
Coordinates of the existing bracings	Automatic detection of bracings
Define manually coordinates of the existing bracings	Add bracings at points where adjoining elements occur
1,20; 2,40; 3,60 m	Add bracings at points where bending moments equal
	zero
Basic scheme of a member	Bracing detection preview For member no.: 1 Belka stęzona
	For load case: 1 c własny -
Buckling coefficients of component segments	<pre>m</pre>
1,00; 1,00; 1,00	
OK Cancel	Нер

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 - Parame	ters - EN 1995-1:2004/A1:2008			
Member type: klas 1,2m góra	i st	Save		
Buckling (Yaxis) Member length ly: Beal Coefficient	Buckling (Z axis) Member length Iz: Real Coefficient	Close Service		
Buckling length coefficient Y:	Buckling lengt <u>h</u> coefficient Z:	<u>M</u> ore <u>O</u> ther		
	Lateral buckling length coeff. Upper flange Id = (Id1,Id2,) Id = Io	<u>Fire</u>		
Method of critical stress determin © Classic - formula (6.31) © For rectangular sections - form	ation - 6.3.3 :	Note		
S <u>e</u> rvice class:	1 •	Help		
	ſ	Definitions -EN	1995-1:2004/A1:2008	
<u>nber</u> of the member m gned to the appropria <i>Member type</i> .		Members Groups	1	New
is very important whe		<u>B</u> ar list: <u>N</u> ame:	1 Belka stęzona	Parameters
erifying different mer	nber types	C. <u>G</u> roup:	<ul> <li>Member type:</li> </ul>	klas 1,2m top 👻

ΟK

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

In the CALCULATIONS dialog set the following:

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

->Configuration.

🗊 Calculations -EN 1995-1:2004/A1:2008				
Verification options				
Member verification:	1to6 11to16 List			
Code group verification:	List			
🔘 Code group <u>d</u> esign:	List			
	Options			
Loads	Limit state			
C <u>a</u> ses: 8	List 🛛 🗸 🛛 🗸			
Calculation archive	List Serviceability			
OK Configuration Calculations Help				

<u>S</u>ave

Help

Before doing calculations you have to remember to specify appropriate duration for loads in the CALCULATIONS dialog:

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

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

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

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

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

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

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

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

### Simplified results tab

RESULTS - Code - EN 1995-1:2	2004/A1:2008		
BALK 75x200		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 Sig_m.z.d = 1.07/187.49 = 5.68		fm.y.d = 11.08 MPa fm.z.d = 12.72 MPa	Eorces Detailed
FACTORS AND ADDITIONAL F km = 0.70 kh = 1.15		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 Z	Пор
RESULTS Sig_m.y.d/fm.y.d + km*Sig_m. Sig_m.y.d/(kcrit*fm.y.d) = 6.36		70°5.68/12.72 = 0.89 <1.00 (6.11) (6.33)	

### Detailed results tab

	<u>A</u> uto	Bar: 1 Point / Coo Load case:	Belka stęzona	ction OK	<ul><li>○</li><li>○</li></ul>	OK
mplified results	Detailed results	:				<u>C</u> han
Symbol	Value	Unit	Symbol description	Section	*	
Sig_m,z,d	5.68	MPa	Left edge normal stress due to Mz	[6.1.6]		
fm,y,d	11.08	MPa	Allowable normal stress from bending	[6.1.6]		
fm,z,d	12.72	MPa	Allowable normal stress from bending	[6.1.6]		<u> </u>
		Fact	ors and additional parameters			Detail
kh	1.15		Scale coefficient	[3.2/3.3/3.4]		
kh_y	1.00		Scale coefficient	[3.2/3.3/3.4]		
kh_z	1.15		Scale coefficient	[3.2/3.3/3.4]		
kl	1.00		Reduction factor depending on member length	[3.4.(4)]		Calc. N
kmod	0.90		Modification factor depending on time of load action	[3.1.3]		Calc. 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 cr	itical stress d	etermination	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)]	Ξ	
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 1995-1:20 ANALYSIS TYPE: Me			
CODE GROUP: MEMBER: 1 Belka stę	zona POINT: 2	COORDINA	<b>TE:</b> $x = 0.50 L = 2.40 m$
LOADS: Governing Load Case: 8	wypadkowe obc		
<b>MATERIAL</b> 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
<b>SECTION PARA</b>	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		<b>ALLOWABLE</b> f m,y,d = 11.08 f m,z,d = 12.72	
Factors and additiona $km = 0.70$ $kh = 1.1$		Ksys = 1.00	
LATERAL BUCK lef = 1.48 m Sig_cr = 70.43 MPa	<b>KLING PARAMETERS:</b> Lambda_rel m = 0.48 k crit = 1.00		
BUCKLING PARAMET	ERS:	About Z axis:	
	ULAS: Sig_m,z,d/f m,z,d = 6.38/11. = 6.38/(1.00*11.08) = 0.58		.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.

		x = 0.50 L = 2.40 m ypadkowe obc	OK
BALK 50x100 BALK 50x115	sults		<u>C</u> hange
BALK 50x125 BALK 50x140	S	ALLOWABLE STRESSES	
BALK 50x150 BALK 50x175 BALK 50x200 BALK 50x225 BALK 50x250 BALK 50x250 BALK 50x250 BALK 50x140 BALK 63x150 BALK 63x150 BALK 63x150 BALK 63x175	8.29 MPa 5.57 MPa	fm.y.d = 12.01 MPa fm.z.d = 13.80 MPa	Eorces Det <u>a</u> iled
BALK 63x200 BALK 63x225	VAL PARAMETERS		
BALK 63x250 BALK 75x150 BALK 75x160 BALK 75x175	1.25 kmod = 0.90	Ksys = 1.00	Calc. Note
BALK 75x200 BALK 75x225	lef = 1.28 m	Lambda_rel m = 0.48	
BALK 75x250	Sig_cr = 68.56 MPa	– k crit = 1.00	Help
BALK 100x200 BALK 100x225		BUCKLING Z	Пар
	"Sig_m,z,d/f m,z,d = 38.29/12.01 + 0.7 t) = 38.29/(1.00*12.01) = 3.19 > 1.00	70*25.57/13.80 = 4.48 > 1.00 (6.11)	

BALK 63x225		/ x = 0.50 L = 2.40 m wypadkowe obc	
Simplified results Detailed result	s		
CALCULATION STRESSES		ALLOWABLE STRESSES	
Sig_m,y,d = 3.19/531.56 = 6.0		fm,y,d = 11.08 MPa	
Sig_m,z,d = 1.07/148.83 = 7.1	6 MPa	f m,z,d = 13.18 MPa	
FACTORS AND ADDITIONAL	PARAMETERS		
km = 0.70 kh = 1.1	9 kmod = 0.90	Ksys = 1.00	
LATERAL BUCKLING	lef = 1.53 m	Lambda_rel m = 0.60	
_ <sup>↓</sup> ↓↓	Sig_cr = 44.37 MPa	– k. crit = 1.00	
BUCKLING Y		BUCKLING Z	51
		$\mathbf{X}$	
RESULTS			=
	n,z,d/f m,z,d = 0.70×6.00/11.0	8 + 7.16/13.18 = 0.92 < 1.00 (6.12)	

The results for the newly selected section are presented below.

TIMBER ST	<b>FRUCTURE CALCULA</b>	ATIONS for BALK 6	3x225
CODE: EN 1995-1:2004/ ANALYSIS TYPE: Memb			
CODE GROUP: MEMBER: 1 Belka stęzon		<b>COORDINATE:</b> $x = 0.50 L$	
LOADS: Governing Load Case: 8 wy			
<b>MATERIAL</b> 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
	AMETERS: BALK 63x225		
ht=22.5 cm bf=6.3 cm	Ay=31.02 cm2	Az=110.78 cm2	Ax=141.80 cm2
tw=3.1 cm tf=3.1 cm	Iy=5980.10 cm4 Wely=531.56 cm3	Iz=468.80 cm4 Welz=148.83 cm3	Ix=1544.5 cm4
<b>STRESSES</b> Sig_m,y,d = MY/Wy= 3.19/5 Sig_m,z,d = MZ/Wz= 1.07/14	48.83 = 7.16 Mpa	f m,y,d = 1 f m,z,d = 1	
Factors and additional pathematical km = $0.70$ kh =	arameters	.90 Ksys = 1.00	
	LING PARAMETERS: Lambda_rel m = 0.6 k crit = 1.00		
BUCKLING PARAMETER About Y axis:	S:	Abo	put Z axis:
Sig_m,y,d/(kcrit*f m,y,d) = 6			(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) $\sigma$ m,cr = f (Leff) - critical bending stress $\sigma$ m,y,d - design bending stress due to My $\sigma$ 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* $\sigma$ 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

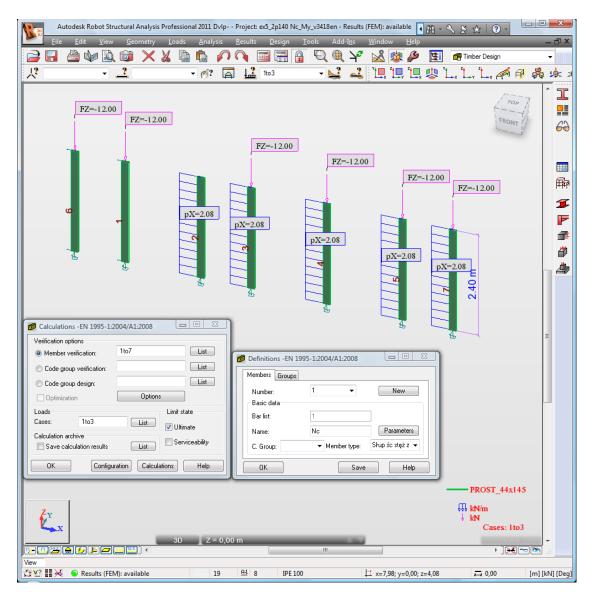
Example based on "Practical design of timber structures to Eurocode 5" Hans Larsen and Vahik Enjily File: EX\_5\_2p140\_Nc\_My.rtd

### 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 19	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	<u>S</u> ave	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 (Y axis) Member length ly: Real	Buckling (Zaxis) Member length Iz:	Close
© Coefficient	<ul> <li>Coefficient</li> </ul>	<u>S</u> ervice
Buckling length coefficient Y:	Buckling length coefficient Z:	More
1,00 I 1.0	1,00 L	<u>0</u> ther
Lateral buckling parameters	<u> </u>	
Lat. buckling type:	Lateral buckling length coeff.	
	ld = lo ld = lo	
Double sections to	eated as solid	
Method of critical stress determina	ation - 6.3.3 :	Note
Classic - 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	×
× II ±	Beam with pinned supports Beam with fixed supports Cantilever Without lateral buckling	OK Cancel Help

select appropriate Load level icon

Doad Level	X
	OK Cancel Help

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

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

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

💋 Load Type		×
	at the end load rated force	OK Cancel Help

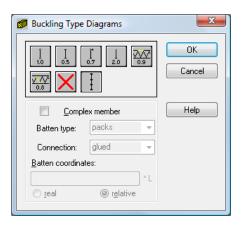
• define bracings for Lateral buckling and Buckling.

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	
	jd = 1,00 lo	
X		
Ŧ	Intermediate bracings	

the second opens > BUCKING TYPE DIAGRAMS dialog.

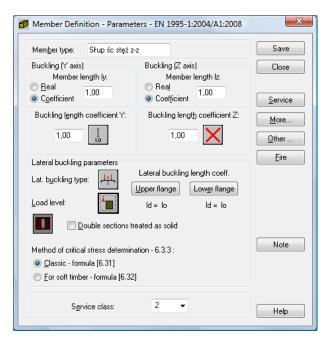


If you click the last icon Intermediate bracings, the new dialog INTERNAL BRACINGS will appear:

🗾 Internal bracings			×
Test for member: 3 Nc+My bez wyb z-z		ng-lower flange	
Buckling Y Buckling Z Lateral buckling-upper flange Lateral	l buckling	-lower flange	
Coordinates of the existing bracings		Automatic detection of bracir	ngs
Define manually coordinates of the existing bracings		Add bracings at points w	here adjoining elements occur
	m	Add bracings at points w	here bending moments equal
		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

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

In CALCULATIONS dialog, set the following:

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

Calculations -EN 1995-1:20	004/A1:2008
<ul> <li>Verification options</li> </ul>	
Member verification:	1to7 List
Code group verification:	List
Code group <u>d</u> esign:	List
	Options
- Loads	Limit state
Cases: 1to3	List 🔽 Ultimate
Calculation archive	Serviceability
Save calculation results	List
OK Configura	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	1 IIII	Re Load Case Classification - Duration
Calculation points Number of points: Characteristic points Calculation parameters Efficiency ratio: 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	OK Cancel Help	Load Case Classification - Duration       Case list:       Number       1       2       wiatr       <
Units of results Code  Robot Camber Take the deflections from the following case into consideration:		

 - 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:

Ru Load Case Classification - Dur	ation					×	
Case list:							
Number Name		Class	Number	Name		ОК	
	>>	1 5	1 2	STA1 wiatr		Cancel	
< III	• <<	<b>ا</b>	1		F	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.

Results Messages	800	- Member Verifi	cation ( UL	S)1to7				Calc. Note Close
Member	Γ	Section	Material	Lay	Laz	Ratio	Case	Help
1 Nc	ЮК	PROST_44x145	C16	57.34	188.95	0.36	1 STA1	
2 My	СК	PROST_44x145	C16	57.34	188.95	0.79	2 wiatr	Ratio
3 Nc+My bez wyb z-z	СК	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	ок	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

Discrete Code - EN 1995-1:	2004/A1:2008		
PROST_44×145 -		/b z-z / x = 0.50 L = 1.20 m KOMB1 (1+2)*1.00	ОК
Simplified results Detailed results	]		<u>C</u> hange
CALCULATION STRESSES Sig_c,0,d = 12.00/63.80 = 1.88 Sig_m,y,d = 1.50/154.18 = 9.73		ALLOWABLE STRESSES f c,0,d = 14.38 MPa f m,y,d = 13.63 MPa	Eorces Detailed
FACTORS AND ADDITIONAL km = 0.70 kh = 1.28 LATERAL BUCKLING		Ksys = 1.00 Lambda_rel m = 0.88 k crit = 0.90	Calc. Note Parameters Help
BUCKLING Y LY = 2.40 m LFY = 2.40 m Lambda Y = 57.34 RESULTS Sig of d//courts of d/d + Sig	Lambda_rel Y = 1.02 ky = 1.10 kcy = 0.67 mud/f mud = 1.89//0.67*14	BUCKLING Z	
Sig_m,y,d/(kcrit*f m,y,d) = 9.7:			

#### Detailed results tab

	<u>A</u> uto ▼		) Nc+My bez wyb z-z rdinate: 2 / x = 0.50 L = 1.20 m 3 KOMB1 (1+2)*1.00	Section OK	<ul><li>○</li><li>○</li></ul>	OK	
implified results	Detailed results	:				<u>Chang</u>	
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]	1		
	,		Buckling parameters		1	Eorce	
			Bucking parameters			Detail	
About the Y a	xis of cross-s	ection				Derail	
LY	2.40	m	Member length	[6.3.2]	]		
LFY	2.40	m	Buckling length	[6.3.2]	]		
Lambda Y	57.34		Member slenderness	[6.3.2]	]		
Sig c,crit,y	16.21	MPa	Critical stress ( buckling )	Critical stress ( buckling ) [6.3.2.(1)]			
Lambda_rel	1.02		Relative slenderness (buckling)	[6.3.2.(1)]	]	C <u>a</u> lc. N	
ky	1.10		Slenderness factor [6.3.2.(3)]				
kcy	0.67 Reduction factor due to compression [6.3.2.(3)]						
			neters of lateral buckling analysis		]	Help	
			n - Classic - formula (6.31)				
lef	2.45	m	Lateral buckling length	[6.3.3]			
Sig_cr	20.79	MPa	Critical stress (lateral buckling)	[6.3.3]			
Lambda_rel	0.88		Relative slenderness (lateral buckling) [6.3.3.(2)]				
k crit	crit 0.90 Lateral buckling factor [6.3.3.(4)]						
			Ratio:				
Delta	0.91		Ratio between normal and allowable stresses	Section OK	] 🛄		
					4		

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: Me			
CODE GROUP: MEMBER: 3 Nc+My b	ez wyb z-z PO	INT: 2 COORDIN	<b>ATE:</b> $x = 0.50 L = 1.20 m$
LOADS: Governing Load Case: 3	KOMB1 (1+2)*1.00		
<b>MATERIAL</b> 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
ht=14.5 cm	AMETERS: PROST_44x1	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
<b>STRESSES</b> Sig_c,0,d = N/Ax = 12.00. Sig_m,y,d = MY/Wy= 1.5		<b>ALLOWABL</b> f c,0,d = 14.33 f m,y,d = 13.6	
Factors and additiona $km = 0.70$ $kh = 1.2$		Ksys = 1.00	
LATERAL BUCK lef = 2.45 m Sig_cr = 20.79 MPa	KLING PARAMETERS: Lambda_rel m = 0.88 k crit = 0.90		
<b>BUCKLING PARAMET</b> $\hat{10}$ 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:	
	ULAS: Sig_m,y,d/f m,y,d = 1.88/(0 = 9.73/(0.90*13.63) = 0.79		= 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
$\begin{array}{l} \lambda y \ \ - \ member \ slenderness \\ ky \ \ - \ slenderness \ factor \\ kcy \ \ - \ reduction \ factor \ due \ to \ compression \\ k_{mod} \\ f \ c,o,d \ \ - \ design \ compression \ strength \\ f \ m,y,d \ \ - \ design \ bending \ strength \ due \ to \ My \\ \sigma \ c,o,d \ \ - \ design \ compression \ stress \\ \sigma \ m,y,d \ \ - \ design \ bending \ stress \ due \ to \ My \end{array}$	[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 <sub>c,y</sub> *fc,o,d) + $\sigma$ m,y,d / f m,y,d =		<u>0,91</u>	<u>0,91</u>

### CONCLUSIONS:

Total agreement of results.