Autodesk[®] Robot[™] Structural Analysis Professional

VERIFICATION MANUAL FOR STEEL MEMBERS DESIGN

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

Government Use

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

RUSSIAN CODE SP 16.13330.2011 MOSCOW 2011	1
NTRODUCTION	2
	2
VERIFICATION PROBLEM 1 DESIGN OF MEMBERS FOR COMPRESSION	6
VERIFICATION PROBLEM 2 LATERAL-TORSIONAL BUCKLING OF BEAMS	15
VERIFICATION PROBLEM 3 COMBINED COMPRESSION AND BENDING ABOUT BOTH AXES	25

Russian code SP 16.13330.2011 Moscow 2011

INTRODUCTION

This verification manual contains numerical examples for elements of steel structures prepared and originally calculated by **Autodesk Robot Structural Analysis Professional version 2015**.

Each problem contains the following parts:

- title of the problem
- specification of the problem
- Robot solution of the problem
- outputs with calculation results and calculation notes

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 required **regional preferences** click Tools / [Preferences...] and in default opened *Preferences* dialog box select in combo boxes a needed country (region) and working / printout language.



B. Job Preferences

To specify required **job preferences** click Tools / [Job Preferences...] and in default opened *Job Preferences* dialog box select preferences corresponding to your structure example at the left list view and appropriate combo boxes. Below there is a screenshot showing the selection [Design codes] :

Job Preferences			? X
Control Contro	EFAULTS Steel/Aluminum structures: Steel connections: Imber structures: RC structures: Geotechnical:	SP 16.13330.2011 EN 1993-1-8:2005/AC:2009 EN 1995-1:2004/A1:2008 SP 63.13330.2012 SNIP 2.02.01-83 More codes	• • •
😤 Open defau	lt parameters		
Save current par	ameters as default	OK Cancel	Help

You can create a new Job Preferences with arbitrarily chosen options and parameters (standards, materials, databases, load codes etc.) appropriate for required condition under a new name to make it easier for future work.

In that case choose proper detailed data from [*Configuration of Code List*] dialog box which is opened after pressing [*More codes...*] button and e.g. looks as shown below :

Configuration of Code List					×
Codes:				Current codes:	
Steel / aluminum		•		Set as current	
Code	Country	*		Code	
SNIP II-23-81	Russia			EN 1993-1:2005/AC:2009	
GP 16.13330.2011	Russia		≥	SNIP II-23-81	
NBE EA-95	Spain			SP 16.13330.2011	
SE-A:2006	Spain				
JNE-EN 1993-1:2008/AC:2009	Spain EC3				
3SK99	Sweden				
ENV 1993-1: 1992 NAD Sweden	Sweden EC3	=			
SS-EN 1993-1:2005/AC:2009	Sweden EC3				
BS 5950:2000	UK	-			
•		•		< III	•
OK Cancel					Help

If you pick Load combinations from Codes combo box the new list view appears:

K Configuration of Code List					x
Codes:				Current codes:	
Load combinations		•		Set as current	
Code	Country	*		Code	
CT 20.13330.2011	Россия			SNiP 2.01.07-85	
СНиП 2.01.07-85	Россия		\geq	SP 20.13330.2011	
ACI318_2002_geo	USA Geotechnic				
ACI	USA Geotechnic				
ASD	USA ASCE 7-05				
LRFD	USA ASCE 7-05				
ACI318	USA				
ACI318_2002	USA				
ASD ASCE 7-10	USA	-			
•		P.		< III	•
OK Cancel					<u>H</u> elp

Set a selected code on the right list of the box using arrows than press it as the current code .

Configuration of Code List						×
Codes:					Current codes:	
Load combinations			•		Set as current	
Code	Country		-		Code	
ACI	USA Geotechnic				SNIP 2.01.07-85	
ACI318	USA			\geq	SP 20.13330.2011	
ACI318_2002	USA			-	СП 20.13330.2011	
ACI318_2002_geo	USA Geotechnic					
AL 76	France					
AL 76 Avril 2000	France					
API/ASD	US					
AS 1170.0:2002	Australia					
AS/NZS 1170.0:2002(4)	Australia		-			
•		•				•
OK Cancel						Help

Press OK.

Job Preferences		? <mark>* × -</mark>)
Image: Constraint of the second s	FAULTS <u>C</u> ode combinations: Snow/ <u>w</u> ind loads: <u>S</u> eismic loads:	C∏ 20.13330.2011 C∏ 20.13330.2011 SP 20.13330.2011 C∏ 20.13330.2011 C∏ 20.107-85 SNP 2.01.07-85 C∏ 20.13330.2011 More More codes
😤 Open defaul	t parameters	
Save current para	meters as default	OK Cancel Help

After the job preferences decisions are set, type a new name in combo box ,e.g.

"new Russian code" and save it pressing Save Job Preferences icon placed on the top of [Job Preferences] dialog box. It opens Save Job Preferences dialog box which you should accept pressing OK.

VERIFICATION PROBLEM 1 design of members for compression

File: SP16_ex_Compression.rtd

TITLE:

Example 1. I-Shape Compressed Column Verification

SPECIFICATION:

Select a C255 (Ry = 240 MPa) I DDSH 40x1 bar to carry an axial dead load of 622 kN and live load of 1800 kN. Assume the design member is 9 m long, is pinned top and bottom in both axes and is laterally braced about the z-z axis at the midpoint. Verify the strength of a defined compression member.



SOLUTION:

You must remember to specify appropriate gm material factor (a new option) in CONFIGURATION dialog box placed at Steel/Aluminium Design level.

In DEFINITIONS dialog box define a new type of member, laterally braced about the *z*-*z* axis at the midpoint . It can be set in *Member type* combo-box.

🗲 Definitions - SP 16.	13330.2011	
Members Groups		
N <u>u</u> mber: Basic data		N <u>e</u> w
<u>B</u> ar list:		Selection
<u>N</u> ame:		Parameters
C. <u>G</u> roup:	✓ Member type:	Simple bar
ОК		Column Beam

Pre-defined type of member "Column" may be initially opened.

For a chosen member type press the *Parameters* button on *Members* tab which opens - here - "*Column*" MEMBER DEFINITION–PARAMETERS dialog box.

ember Definition - Parameters -	SP 16.13330.2011	×
Member type: Column		Save
Buckling (y axis) Member length lef,y: <u>R</u> eal <u>C</u> oefficient	Buckling (z axis) Member length lef,z: Real © Coefficient	Close Service
Buckling length coefficient Y:	Buckling length coefficient Z:	<u>M</u> ore
Section type Y: auto	Section type Z: auto ▼	Stiffeners
Elexural - torsional buckling for mono-symmetrical I sections.	tees, chanels and	
Lateral buckling parameters		
Lateral buckling	Upper flange	
Load level:	lef,b = lo	
Load type:	lef,b = lo	Help

Type a new name in the *Member type* editable field. Change parameters to meet initial data requirements of a structure in a current task. In this particular compression case define buckling z-z parameters. Press *Buckling length coefficient Z* icon which opens BUCKLING DIAGRAMS dialog box.



Click a last icon.

The new dialog box *INTERNAL BRACING* will appear with active *Buckling Z* tab . In *Buckling Z* tab define internal support in the middle of the member by typing relative value 0.5 for marked *Define manually coordinates of the existing bracings* field.



Press OK.

Save the newly-created member type under a new name, e.g. "0,5 Lz bracing" :

Member Definition - Parameters -	- SP 16.13330.2011	×
Member type: 0,5 Lz bracing		Save
Buckling (y axis) Member length lef, y: Real	Buckling (z axis) Member length lef,z:	Close
Oefficient	Operficient	Service
Buckling length coefficient Y:	Buckling length coefficient Z:	<u>M</u> ore
Mi y = 1.00	Mi z = Auto	Stiffeners
Section type Y: auto ▼	Section type Z: auto 🔻	
Elexural - torsional buckling for mono-symmetrical I sections.	r tees, chanels and	
Lateral buckling parameters		
Lateral buckling	Lateral buckling length coeff.	
Load level:	lef,b = lo	
Load type:	Lower flange	Help

Number of the member must be	📁 Definitions - SF	P 16.13330.2011	x
assigned to appropriate name	Members Group	DS	
of Member type.	Number:	1 ▼ <u>Ne</u> w	
(It is very important when you verify	Basic data		
different member types.)	<u>B</u> ar list:	1	
	<u>N</u> ame:	Column_1 Parameters	
	C. <u>G</u> roup:	✓ Membertype: 0,5 Lz bracing ▼	

OK

Save

Help

In the CALCULATIONS dialog box set for this task:

5 Calculations - SP 16.13330.2011	Configuration - SP 16.13330.2011
Verification options	Calculation points
<u>Member verification:</u>	Number of points: 7
Code group verification:	Cancel
Code group design:	Calculation parameters Effigiency ratio: 1.00 Help Components of complex bars are not taken into account
Cases: 1to4 List VUtimate	Code parameters Structure importance factor gn:
Save calculation results	gm=1.025 Material factor: gm=1.050
OK Configuration Calculations Help	Calculations taking plastic redistribution into consideration
• Verification option \rightarrow Member Verification	✓ Limit <u>s</u> lenderness: Define
• Loads cases \rightarrow 1 to 4	Compression: Lam,max = 180 - 60*alfa
 Limit state → only Ultimate Limit state will be analyzed so switch off Limit stat Serviceability 	Tension: Lam,max = 250 Internal forces taken into consideration Forces: V Fx V Fy V Fz
 Material factor → gm =1,025; the "1,025" radio button is switched on by default in CONFIGURATION dialog box (opened by [Configuration] button) 	Moments: Mx Wy Mz Units of results Code
 Limit slenderness → for "Main columns" on [Compression] tab ; radio button selected in LIMIT SLENDERNESS VALUE dialog box which is opened by [Define] button placed in CONFIGURATION dialog box 	Take deflections from the following case into consideration: 1 DL1

ompression Tension		ОК
STRUCTURE ELEMENTS	LIMIT SLENDERNESS	Cancel
1. Chords, cross-braces and support diagonals transmitting support reactions		Help
 a) 2D frames, structural and 3D constructions made of pipes and double angles, up to 50 m altitude. 	🔘 180 - 60*alfa	
b) 3D frames made of single angles, 3D frames made of pipes and double angles, above 50 m altitude.	© 120	
 Elements not included in & 1 and 7 a) 2D frames, welded structural and 3D constructions made of single angles, structural and 3D constructions made of pipes and double angles. 	© 210 - 60*alfa	
b) 3D and structural constructions made of single angles with bolted connections.	© 220 - 40*alfa	
3. Upper truss chords not fixed during assembly (after completing the assembly assume limit slenderness as in $\&$ 1)	© 220	
4. Main columns	180 - 60*alfa	
 Secondary columns, elements of truss columns, elements of vertical stiffeners between columns (under crane girders) 	🔘 210 - 60*alfa	
 Stiffener elements not included in & 5, beams limiting buckling length of compressed elements and other unloaded elements not included in & 7. 	© 200	
7. Compressed and unloaded elements of 3D structures made of T-sections and cross-shaped sections exposed to wind action with the check of slenderness in the vertical	© 150	
	0 200.00	

Now, start calculations by pressing *Calculations* button.

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

1	SP 16.13330.20	11 - Member Veri	fication (ULS)	1				
ſ	Results Message	S						Calc. Note Close
J.	Member	Section	Material	Lay	Laz	Ratio	Case	Help
L	1 Column_1	K ДДШ 40х1	C255	52.72	61.22	0.90	3 COMB1	Patio
								Analysis Map
L								Calculation points
L								Division: n = 7
								Additional: none

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

Simplified results tab

FRESULTS - Code - SP 16.13330.2011		
ДДШ 40x1 ▼ Load case:	In_1 1 /x = 0.00 L = 0.00 m 3 COMB1 1*1.00+2*1.20	ОК
Simplified results Detailed results		Change
INTERNAL FORCES AND FACTORS N = 2793.0 kN		
STRESSES IN THE ANALYZED SECTION SigN = 175.66 MPa		Forces
BUCKLING y	BUCKLING z	Calc Note
iv = 9.00 m Fiy = 0.86 lefy = 9.00 m Lamy = 52.72 Lam_y = 1.80	Iz = 9.00 m Fiz = 0.81 lefz = 4.50 m Lamz = 61.22 Lam_z = 2.09	Hala
SECTION CHECK Web:Lam_w/Lam_uw = 0.49 < 1.0; Flange:Lam_f N/(A*Ry*gc1/gn) = 0.73 < 1.00 [7.1.1-(7)]	i/Lam_uf = 0.40 < 1.0 [7.3] COMPACT SECTION	Trep
MEMBER STABILITY CHECK Lamy = 52.72 < Lamy.max = 128.64 Lamz = 61 N/(min(Fiy.Fiz)*A*Ry*gc2/gn) = 0.90 < 1.00 [7.1-(7)]	.22 < Lamz,max = 125.93 [10.4.1]	

Detailed results tab

RESULTS - Cod	e - SP 16.13330.2	2011				X
ДДШ 40x1 Simplified results	Auto	Bar: Point / Coo Load case:	1 Column_1 rdinate: 1 / x = 0.00 L = 0.00 m 3 COMB1 1*1.00+2*1.20	Section OK	• • •	OK Change
Symbol	Value	lloit	Symbol description	Section	•	
SigN	175.66	MPa	Normal stress resulting from axial force	[7.1.1]		
About the via	vie of cross_sec	E	uckling analysis parameters			<u>F</u> orces
Type	b		Type of cross-section under compression	ITable 71		
ly	9.00	m	Theoretical member length	[10]		
lefy	9.00	m	Buckling length	[10]		
Lamy	52.72		Member slenderness	[10]		
Lamy,max	128.64		Maximum member slenderness	[Table 32]		Calc. Note
Lam_y	1.80		Relative member slenderness	[7.1.3]		
Fiy	0.86		Buckling factor	[7.1.3]		
About the z a	xis of cross-sec	tion				
Туре	b		Type of cross-section under compression	[Table 7]	_	Help
lz	9.00	m	Theoretical member length	[10]		
lefz	4.50	m	Buckling length	[10]	=	
Lamz	61.22		Member slenderness	[10]	-	
Lamz,max	125.93		Maximum member slenderness	[Table 32]		
Lam_z	2.09		Relative member slenderness	[7.1.3]		
Fiz	0.81		Buckling factor	[7.1.3]		
			Verification formulas		-	

Pressing the [*Calc.Not*e] button in "RESULTS –Code" dialog box 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.

a) In the first calculation step DDSH 40x1 section was considered. The results are presented below.

STEEL DESIGN					
CODE: SP 16.13330.20 ANALYSIS TYPE: Mem	011* Steel structures. ber Verification				
CODE GROUP: MEMBER: 1 Column_1	POINT: 1	COORDINA	TE: x = 0.00 L = 0.00 m		
LOADS: Governing Load Case: 3	COMB1 1*1.00+2*1.20				
MATERIAL C255 Ry = 240.00 MPa gu = 1.30	Rs = 139.20 MPa gc1 = 1.00	Ru = 360.00 MPa gc2 = 1.00	E = 206000.00 MPa gn = 1.00		
SECTION PARA	AMETERS: ддш 40x1				
bf=30.2 cm	Ay=112.95 cm2	Az=45.72 cm2	Ax=159.00 cm2		
t=1.1 cm	Iy=46330.00 cm4	Iz=8590.00 cm4	Ix=151.81 cm4		
tf=1.9 cm	Wely=2330.48 cm3	Welz=568.87 cm3			
Web: hef/t = 27.50 Flange: bef/tf = 6.59	Lam,uw = 56.58 Lam,uf = 16.67				
STRESSES AT CHARA SigN = 175.66 MPa	CTERISTIC SECTION P	OINTS			
	LING PARAMETERS:				
BUCKLING PARAMETE build About Y axis: 1y = 9.00 m 1efy = 9.00 m Lamy = 52.72 $Lam_y = 1.80$	RS: Fiy = 0.86	About Z axis: 1z = 9.00 m 1efz = 4.50 m Lamz = 61.22 $Lam_z = 2.09$	Fiz = 0.81		
VERIFICATION FORMU Section check Web: Lam_w/Lam_uw = 0. N/(A*Ry*gc1/gn) = 0.73 <	LAS: 49 < 1.0; Flange: Lam_f/Ι 1.00 [7.1.1-(5)]	.am_uf = 0.40 < 1.0 [7.3]	COMPACT SECTION		
<u>Member stability check</u> Lamy = 52.72 < Lamy,max = N/(min(Fiy,Fiz)*A*Ry*gc)	= 128.64 Lamz = 61.22 2/gn) = 0.90 < 1.00 [7.1-(7)	< Lamz,max = 125.93 [10.4.1]]		
Section OK !!!					

b) From economical reason try to check a lighter I-section.

Being still in RESULTS- CODE dialog box, delete number of DDSH or type DB section symbol in the editable field below a drawing of a section and select DB55x2. Press ENTER. Calculations (and results) are refreshed instantly.

1	RESULTS - Code - SP 16.13330.2011		
	Bar: 1 Column_1	ОК	
	ДБ 18x2 Load case: 3 COMB1 1*1.00+2*1.20 ДБ 20x1 дБ 2x1 JL5 2x1 ДБ 28x1 JL5 28x2 FACTORS	Qhange	
	ДБ 30x1 ДБ 30x2 ДБ 35x2 ДБ 45x1 ДБ 40x2 ДБ 40x1 ДБ 45x1 ДБ 45x2	Forces	
	∐b 50x1 Дb 50x2 Дb 55x1 Дb 55x2 Дb 56x2 Дb 56x2 Дb 56x2 Дb 50x2 Дb 50x1 Дb 50x1 Дb 50x1 Дb 50x1 Дb 50x1 Дb 50x1 Дb 50x1	Calc. Note	
🗲 RESULTS - Co	ode - SP 16.13330.2011		x
ДБ 70x1	Auto Bar: 1 Column_1 Incorrect section Point / Coordinate: 1 / x = 0.00 L = 0.00 m Load case: 3 COMB1 1*1.00+2*1.20	n	ок
Simplified result	s Detailed results		<u>Q</u> hange
N = 2793.0 k	FORCES AND FACTORS N		
STRESSES SigN = 169.5	IN THE ANALYZED SECTION 8 MPa		<u>F</u> orces
	UCKLING		
BUCKLING	BUCKLING z		Cala Nata
lef 1.0 lef La	= 9.00 m Fiy = 0.96 Iz = 9.00 m Fiz = 0.66 y = 9.00 m mmy = 32.55 Lamz = 85.56 m_y = 1.11 Lam_z = 2.92		
SECTION CI Web: Lam_w N/(A*Ry*gc1	HECK /Lam_uw = 0.78 < 1.0; Flange: Lam_f/Lam_uf = 0.34 < 1.0 [7.3] COMPACT SECTION /gn) = 0.71 < 1.00 [7.1.1-(7)]		Нер
MEMBER S Lamy = 32.5 N/(min(Fiy,Fiz	TABILITY CHECK 5 < Lamy.max = 135.84 Lamz = 85.56 < Lamz.max = 120.00 [10.4.1] z)*A*Ry*gc2/gn) = 1.07 > 1.00 [7.1-(7)]		

The results in the form of a "Calculation Note" for the new selected section are presented below.

STEEL DESIGN

CODE: SP 16.13330.20 ANALYSIS TYPE: Mem	11* Steel structures. ber Verification		
CODE GROUP: MEMBER: 1 Column_1	POINT: 1	COORDINAT	E: $x = 0.00 L = 0.00 m$
LOADS: Governing Load Case: 3	COMB1 1*1.00+2*1.20		
MATERIAL C255 Ry = 240.00 MPa gu = 1.30	Rs = 139.20 MPa gc1 = 1.00	Ru = 360.00 MPa gc2 = 1.00	E = 206000.00 MPa gn = 1.00
	AMETERS: дБ 70x1		
ht=09.1 cm bf=26.0 cm t=1.2 cm tf=1.6 cm Web: Flange:	Ay=80.60 cm2 Iy=125930.01 cm4 Wely=3644.86 cm3 hef/t = 51.00 bef/tf = 6.45	Az=82.92 cm2 Iz=4556.00 cm4 Welz=350.46 cm3 Lam,uw = 65.10 Lam,uf = 19.10	Ax=164.70 cm2 Ix=104.35 cm4
INTERNAL FORCES AN N = 2793.0 kN STRESSES AT CHARAG SigN = 169.58 MPa	D FACTORS	POINTS	
	LING PARAMETERS:		
BUCKLING PARAMETE About Y axis: 1y = 9.00 m lefy = 9.00 m Lamy = 32.55 Lam_y = 1.11	RS: Fiy = 0.96	About Z axis: 1z = 9.00 m 1efz = 4.50 m Lamz = 85.56 Lam_z = 2.92	Fiz = 0.66
VERIFICATION FORMU <u>Section check</u> Web: Lam_w/Lam_uw = 0. N/(A*Ry*gc1/gn) = 0.71 < <u>Member stability check</u> Lamy = 32.55 < Lamy,max = N/(min(Fiy,Fiz)*A*Ry*gc2	LAS: 78 < 1.0; Flange: Lam_f/ 1.00 [7.1.1-(7)] = 135.84 Lamz = 85.56 2/gn) = 1.07 > 1.00 [7.1-(7)]	Lam_uf = 0.34 < 1.0 [7.3] CO i < Lamz,max = 120.00 [10.4.1])]	MPACT SECTION

Incorrect section !!!

VERIFICATION PROBLEM 2 Lateral-torsional buckling of beams

File: SP16_ex_LTB beam.rtd

TITLE:

Example 2. I-Shape Flexural Member Design in Strong-Axis Bending Braced at Midspan

SPECIFICATION:

Verify the strength of the C345 I DB50x1 beam with a simple span of 10.67m. The beam is braced at the ends and center point. The nominal loads are a uniform dead load of 5,0 kN/m and a uniform live load of 8,0 kN/m.

工品 画師 Fピ パー・ユー・パ 国 🗳 3:KOMB_ULS ・	🚅 🇊 Internal bracings 📃 🛋
	Lateral buckling-upper flange
$\begin{tabular}{ c c c c c c c c c c c c c c c c c c c$	Test for member: 1 braced beam 2 × Budday Y Budday Z Lateral budding-upper flarge Lateral budding lower flarge Define sognets budding Z Lateral budding lower flarge Budday Contracts of the easting budding lower flarge Define sognets budden of the easting budding Budding of flores of component segments Budding of flores of component segments Define sognets budden of the easting budding *L Budding defection pervise Ad automatically conditioned of binsching Budding defection pervise For member no: Ad automatically conditioned are located To start where informal nodes are located To start where informal nodes are located if all points where informal nodes are located *L *L
Zerkier unform bad 12 Pr-0.0 Pr-0.5 P	

SOLUTION:

In DEFINITIONS dialog box define a new type of member, torsional and laterally braced upper flange about the *z-z* axis at the midpoint. It can be set in *Member type* combo-box.

Definitions - SP 16.1	3330.2011	
Members Groups		
N <u>u</u> mber: ∂Basic data		New
<u>B</u> ar list:		Selection
<u>N</u> ame:		Parameters
C. <u>G</u> roup:	▼ Member type:	Cincile has
ОК		Column Beam

Pre-defined "beam" member type may be initially opened.

For chosen member type press the [*Parameters*] button on *Members* tab. It opens MEMBER DEFINITION–PARAMETERS dialog box.

		- 3P 10.13330.2011	
Member type: B	eam		Save
Buckling (y axis) Member length Real <u>C</u> oefficient	lef,y: 00	Buckling (z axis) Member length lef,z: Real Coefficient	Close <u>S</u> ervice
Buckling length coe Mi y = 1.00	fficient Y:	Buckling length coefficient	Z: More Sti <u>f</u> feners
Section type Y:	auto 👻	Section type Z: auto	•
Lateral building and	risecuons.		
Lateral buckling		Lateral buckling length coeff. Upper flange lef,b = lo Lower flange	

Type a new name in the *Member type* editable field. Then, change parameters to meet initial data requirements of the structure. In this particular bending case set the following lateral-buckling parameters :

- switch on Lateral buckling
- select upper load level by pressing proper icon



define appropriate load type by pressing " load " icon which opens LOAD TYPE dialog box;

Load type	- SP 16.13330.2011	×
	Uniform loads	ок
-	Concentrated force in center	Help
1/41 ↔	Concentrated force at 1/4 of span leng	th
1	Concentrated force at the end of cantile	ever
	Uniform on cantilever	
	Simple bending	

for this task the first icon "uniform loads" was selected.

• define adequate bracings for Lateral buckling.

To define *Lateral buckling length coefficient* for this structure press *Upper flange* button. It opens LATERAL BUCKLING LENGTH COEFFICIENTS dialog box.



Click the last icon Intermediate bracings. The new dialog box INTERNAL BRACINGS will appear with automatically active Lateral buckling - Upper flange tab.

In *INTERNAL BRACINGS* dialog box there are possibilities of defining independent bracings for buckling and lateral buckling of the marked *member type*.

In *Lateral buckling-upper flange* tab define internal support in the middle of the member by typing typing relative value 0.5 for marked *Define manually coordinates of the existing bracings* field.



Press OK.

By pressing [Service] button at MEMBER DEFINITION–PARAMETERS dialog box open SERVICEABILITY-DISPLACEMENT VALUES dialog box in which limit dispalcements should be defined.

Serviceability - Displacement Values - SP 16.13330.2 🗾 🎫				
Limit displacements Member deflection (local system) Final deflection $\underline{y} = L / 200.(\underline{z} = L / 360.(\underline{v})$ \underline{C} antilever Node displacements (global system) $\underline{x} = L / 150.(\underline{v} = L / 150.(\underline{v})$	OK Cancel Help			
Members with camber Control of displacements with camber considered User-defined camber Uy = 0.0 cm uz = 0.0 cm Automatic camber (additional parameters may be found in the dialog box for calculation configuration)				

Save the newlycreated member type under a new name , e.g. as "LB 0,5l up"

ember Definition - Parameters	- SP 16.13330.2011	<u> </u>
Member type: LB 0,5l_up		Save
Buckling (y axis) Member length lef,y: <u>R</u> eal <u>C</u> oefficient	Buckling (z axis) Member length lef,z: Real Coefficient	Close Service
Buckling length coefficient Y: Mi y = Auto	Buckling length coefficient Z: Mi z = Auto	More Stiffeners
Section type Y: auto ▼ Elexural - torsional buckling for mono-symmetrical I sections.	Section type Z: auto ▼ r tees, chanels and	
Lateral buckling parameters Lateral buckling Load level:	Lateral buckling length coeff. Upper flange lef,b = (lef,b1,lef,b2,) Lower flange	
	lef,b = lo	Help

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

It is very important when you verify different member types.

🗲 Definitions - SP	16.13330.2011	
Members Groups]	
N <u>u</u> mber:	1 🔹	New
Basic data		
<u>B</u> ar list:	1	
<u>N</u> ame:	braced beam	Parameters
C. <u>G</u> roup: 1	✓ Member type:	LB 0,51_up
ОК	<u>S</u> ave	Help

In the CALCULATIONS dialog box set for this task :

5 Calculations - SP 16.13330.2011	Configuration - SP 16.13330.2011
Verification options	Calculation points
Member verification: 12 List	Number of points: 3
Code group verification: 12	Cancel
○ Code group design: 1 List	Calculation parameters
Optimization Options	Efficiency ratio: 1.00
Loads Limit state	Components of complex bars are not taken into account
C <u>a</u> ses: 3 11 List ♥ Ultimate	Code parameters
Calculation archive	factor gn:
Save calculation results	gm=1.025 Material factor:
OK Configuration Calculations Help	◎ gm=1.050
	Calculations taking plastic redistribution into consideration
• Verification option \rightarrow Member Verification	✓ Limit <u>s</u> lenderness: Define
• Loads cases \rightarrow n° 3 and n° 11	Compression: Lam,max = 200
• Limit state \rightarrow Ultimate and Serviceability Limit state	Tension: Lam,max = 250
will be analyzed so switch on both check boxes	Internal forces taken into consideration
• Material factor \rightarrow gm =1,025;	Forces: V Fx V Fy V Fz
the "1,025" radio button is switched on by default in	Moments: W Mx W My W Mz
CONFIGURATION dialog box (opened by [Configuration] button)	□ Units of results ○ <u>C</u> ode
Limit clandernase - "Defined by the user" or	Camber
[Compression] tab (radio button selected in LIMIT	Take deflections from the following case into consideration:
SLENDERNESS VALUE dialog box which is	1 STA1 -
opened by [Define] button placed in	
CONTIGUIATION UIAIOY DUX)	

Tension		ОК
STRUCTURE ELEMENTS	LIMIT SLENDERNESS	Cancel
1. Chords, cross-braces and support diagonals transmitting support reactions		Help
 a) 2D frames, structural and 3D constructions made of pipes and double angles, up to 50 m altitude. 	🔘 180 - 60*alfa	
b) 3D frames made of single angles, 3D frames made of pipes and double angles, above 50 m altitude.	© 120	
2. Elements not included in & 1 and 7		
 a) 2D frames, welded structural and 3D constructions made of single angles, structural and 3D constructions made of pipes and double angles. 	🔘 210 - 60*alfa	
b) 3D and structural constructions made of single angles with bolted connections.	🔘 220 - 40*alfa	
3. Upper truss chords not fixed during assembly (after completing the assembly assume limit slenderness as in & 1)	© 220	
4 Main columns	🔘 180 - 60*alfa	
	@ 210 60*alfa	
 Secondary columns, elements of truss columns, elements of vertical stiffeners between columns (under crane girders) 	0 210 - 00 · alla	
5. Secondary columns, elements of truss columns, elements of vertical stiffeners between columns (under crane girders) 6. Stiffener elements not included in & 5, beams limiting buckling length of compressed elements and other unloaded elements not included in & 7.	© 200	
5. Secondary columns, elements of truss columns, elements of vertical stiffeners between columns (under crane girders) 6. Stiffener elements not included in & 5, beams limiting buckling length of compressed elements and other unloaded elements not included in & 7. 7. Compressed and unloaded elements of 3D structures made of T-sections and cross-shaped sections exposed to wind action with the check of slenderness in the vertical	© 200 © 150	

Now, start verifications by pressing [Calculations] button.

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

1	🎜 SP 16.13330.2011 - Member Verification (SLS ; ULS) 1 2										- O X
ſ	Results Messages									C <u>a</u> lc. Note	Close
Ш	Member	Section	Material	Lay	Laz	Ratio	Case	Ratio(uz)	Case (uz)		Help
	1 braced beam	K ДБ 50х1	C345	53.36	256.69	0.93	3 KOMB_ULS	0.97	11 KOMB_SLS	Dette	
Ш	2 NOT braced beam	🔀 ДБ 50x1	C345	53.36	256.69	3.30	3 KOMB_ULS	0.97	11 KOMB_SLS	natio	
										Analysis	Мар
L										Calculation po	pints
Ш										Division:	n = 3
										Extremes:	none
										Additional:	none
L											

Pressing the line with general results for the member 1 opens the RESULTS dialog box with detailed results for the analyzed member. The view of the RESULTS windows are presented below.

Simplified results tab

Auto Bar: 1 braced beam Section OK Image $JL5 50x1$ Point / Coordinate: $2/x = 0.50 L = 5.33 m$ Load case: $Change$ Simplified results Displacements Detailed results INTERNAL FORCES AND FACTORS My = 267.44 kN'm Image STRESSES IN THE ANALYZED SECTION SigMy = 177.05 MPa Eorces LATERAL BUCKLING Image Image IMID Image File = 5.33 m Psi = 2.52 Alpha = 3.86 Fil = 0.60 Fib = 0.60 BUCKLING y BUCKLING z Calc. Note Image Image Image Image SECTION CHECK Web: Lam_w/Lam_uw = 0.60 < 1.0; Range: Lam_f/Lam_uf = 0.36 < 1.0 [8.5] COMPACT SECTION My/(Wyn.minTky'gc1/gn) = 0.55 < 1.00 [8.2.1-(41)] MEMBER STABILITY CHECK Help	F RESULTS - Code - SP 16.13330.2011	- 🗆 🗙
Simplified results Change INTERNAL FORCES AND FACTORS My = 267.44 kN*m Eorces STRESSES IN THE ANALYZED SECTION SigMy = 177.05 MPa Eorces LATERAL BUCKLING Eorces Image Image Image Eorces BUCKLING y Eorces BUCKLING y BUCKLING z Image Image SECTION CHECK Eorces Web: Lam_w/Lam_uw = 0.60 < 1.0; Flange: Lam_f/Lam_uf = 0.36 < 1.0 [8.5]	Auto Bar: 1 braced beam Point / Coordinate: 2 / x = 0.50 L = 5.33 m Load case: 3 KOMB_ULS 1*1.20+2*1.60	ОК
INTERNAL FORCES AND FACTORS My = 267.44 kN*m STRESSES IN THE ANALYZED SECTION SigMy = 177.05 MPa LATERAL BUCKLING Efb = 5.33 m Psi = 2.52 Image: Appha = 3.86 Fit = 0.60 Fib = 0.60 BUCKLING y BUCKLING z Cgic. Note SECTION CHECK SECTION CHECK Help Web: Lam_w/Lam_uw = 0.60 < 1.0; Flange: Lam_f/Lam_uf = 0.36 < 1.0 [8.5] COMPACT SECTION	Simplified results Displacements Detailed results	Change
STRESSES IN THE ANALYZED SECTION Eorces SigMy = 177.05 MPa Eorces LATERAL BUCKLING Psi = 2.52 H Image: Lam (b) = 5.33 m Apha = 3.86 Fi1 = 0.60 BUCKLING y BUCKLING z SECTION CHECK SECTION CHECK Web: Lam w/Lam uw = 0.60 < 1.0; Flange: Lam (f/Lam uf = 0.36 < 1.0 [8.5] COMPACT SECTION	INTERNAL FORCES AND FACTORS My = 267.44 kN*m	
LATERAL BUCKLING Image: Lam grad backgrad back	STRESSES IN THE ANALYZED SECTION SigMy = 177.05 MPa	Forces
BUCKLING y BUCKLING z Calc. Note SECTION CHECK Web: Lam_w/Lam_uw = 0.60 < 1.0; Flange: Lam_f/Lam_uf = 0.36 < 1.0 [8.5] COMPACT SECTION	LATERAL BUCKLING Lift, b = 5.33 m Psi = 2.52 Alpha = 3.86 Fi1 = 0.60 Fib = 0.60	
SECTION CHECK Help Web: Lam_w/Lam_uw = 0.60 < 1.0; Flange: Lam_f/Lam_uf = 0.36 < 1.0 [8.5] COMPACT SECTION	BUCKLING z	Calc. Note
MEMBER STABILITY CHECK	SECTION CHECK Web: Lam_w/Lam_uw = 0.60 < 1.0; Flange: Lam_f/Lam_uf = 0.36 < 1.0 [8.5] COMPACT SECTION My/(Wyn,min*Ry*gc1/gn) = 0.55 < 1.00 [8.2.1-(41)]	Help
My/(Fib*Wcy*Fy*gc2/gn) = 0.93 < 1.00 [8.4.1-(69)]	MEMBER STABILITY CHECK My/(Fib*Wcy*Ry*gc2/gn) = 0.93 < 1.00 [8.4.1-(69)]	

Displacements results tab

ﷺ RESULTS - Code - SP 16.13330.2011	- • ×
Auto Bar: 1 braced beam Section OK	ОК
Simplified results Displacements Detailed results Bar deflection	Change
uz = 2.9 cm < uz max = L/360.00 = 3.0 cm Verified Governing load case: 11 KOMB_SLS (1+2)*1.00	Forces
Node displacements Not analyzed	Calc. Note
	Help

Detailed results tab

RESULTS - Cod	e - SP 16.13330	.2011					
16 50x1	<u>A</u> uto	Bar: 1 Point / Coord Load case:	braced beam dinate: 2 / x = 0.50 L = 5.33 m 3 KOMB_ULS 1*1.20+2*1.60	ОК	。 。	ОК	
implified results	Displacements	Detailed resu	its]	<u>C</u> hange	
Symbol	Value	Unit	Symbol description	Section			
Lam,uw	81.19		Allowable web slenderness	[8.5.1]	-		
Lam_w	1.91		Relative web slenderness	[8.5.1]			
Lam_uw	3.20		Relative allowable slenderness of a web	[8.5.1]		Forces	
Flange							
bef	7.5	cm	Effective flange width	[7.3.1]			
tf	1.2	cm	Flange thickness	[7.3.1]			
bef/tf	6.22		Effective flange slenderness	[8.5.18-19]			
Lam,uf	17.06		Allowable flange slenderness	[8.5.18-19]			
Lam_f	0.25		Relative flange slenderness	[8.5.18-19]		Calc. Note	
Lam_uf	0.67		Relative allowable slenderness of a flange	[8.5.18-19]			
	Internal forces and factors						
Му	267.44	kN*m	Bending moment My			Help	
Betay	1.00		coef. for calculations using method of plastic deformatio	[8.2.3-(52)]			
SigMyw	177.05	MPa	Stress resulting from My moment at the upper edge	[8.2.1]			
SigMyn	-177.05	MPa	Stress resulting from My moment at the lower edge	[8.2.1]			
		L	ateral buckling parameters		-		
•			III	- F			

Pressing the *[Calc.Note]* button in "RESULTS –Code" dialog box 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.

STEEL DESIGN							
CODE: SP 16.13330.2011* Steel structures. ANALYSIS TYPE: Member Verification							
CODE GROUP: MEMBER: 1 braced bear	CODE GROUP: MEMBER: 1 braced beam POINT: 2 COORDINATE: x = 0.50 L = 5.33 m						
LOADS: Governing Load Case: 3	KOMB_ULS 1*1.20+2*1.60)					
MATERIAL C345 Ry = 320.00 MPa gu = 1.30	Rs = 185.60 MPa gc1 = 1.00	Ru = 460.00 MPa gc2 = 1.00	E = 206000.00 MPa gn = 1.00				
SECTION PAR	AMETERS: дБ 50x1						
bf=20.0 cm	Ay=48.00 cm2	Az=43.30 cm2	Ax=92.98 cm2				
t=0.9 cm	Iy=37160.00 cm4	Iz=1606.00 cm4	Ix=34.22 cm4				
tf=1.2 cm	Wely=1510.57 cm3	Welz=160.60 cm3					
Web:	hef/t = 48.41	Lam,uw = 81.19					
Flange:	bei/u = 0.22	Lam,ur = 17.00					
INTERNAL FORCES AN	ID FACTORS My = 267.44 kN*m						
STRESSES AT CHARA	CTERISTIC SECTION PC SigMy = 177.05 MPa	DINTS					
LATERAL I	BUCKLING PARAMETER	lS:					
Alpha = 3.86	Fi1 = 0.60	Fib = 0.60					
BUCKLING PARAMETE About Y axis:	RS:	About Z axis:					
Section check Web: Lam_w/Lam_uw = 0.60 < 1.0;							
<u>Member stability check</u> My/(Fib*Wcy*Ry*gc2/gn) = 0.93 < 1.00 [8.4.1-(69)]							
	S						
Deflections							
uz = 2.9 cm < uz max = L/3 Governing Load Case: 1	660.00 = 3.0 cm 1 KOMB_SLS (1+2)*1.00	Verified					
Displacements Not analyzed							

Section OK !!!

For comparison the simplified results for the unbraced beam, $n^{\circ} 2$, are presented below.

Simplified results tab

FRESULTS - Code - SP 16.13330.2011	X
Auto Bar: 2 NOT braced beam □ДБ 50x1 ▼ Incorrect section Incorrect section □	ОК
Simplified results Displacements Detailed results	<u>C</u> hange
INTERNAL FORCES AND FACTORS My = 267.44 kN*m	
STRESSES IN THE ANALYZED SECTION SigMy = 177.05 MPa	<u>F</u> orces
Image: Apple a 10.67 m Psi = 2.33 Alpha = 15.43 Fi1 = 0.17 Fib = 0.17	
BUCKLING z	C <u>a</u> lc. Note
SECTION CHECK Web:Lam_w/Lam_uw = 0.60 < 1.0; Flange:Lam_f/Lam_uf = 0.36 < 1.0 [8.5] COMPACT SECTION My/(Wyn.min*Ry*gc1/gn) = 0.55 < 1.00 [8.2.1-(41)]	Help
MEMBER STABILITY CHECK My/(Fib*Wcy*Ry*gc2/gn) = 3.30 > 1.00 [8.4.1-(69)]	

VERIFICATION PROBLEM 3 combined compression and bending about both axes

File: SP16_ex_Nc_My_Mz.rtd

TITLE:

Example 3. I-shape Subjected to Combined Compression and Bending About Both Axes

SPECIFICATION:

Verify if an C275 I DSZ_50x1 bar has sufficient available strength to support the axial forces and moments listed below. The unbraced length is 7,70m and the member has pinned ends: Lefy = Lefz = Lef, b = 7,70 m

section internal forces	Material	Properties:
Nc = 800 kN $My = 400 kNm$ $Mz = 30 kNm$	C275	R <i>y</i> = 270 MPa , R <i>u</i> = 370 MPa



SOLUTION:

For a considered task define manually a new type of member in DEFINITIONS dialog box. It can be set in *Member type* combo-box.

🗲 Definitions - SF	16.13330.2011	
Members Group	S	
N <u>u</u> mber: Basic data	1 •	New
<u>B</u> ar list:	1	
<u>N</u> ame:	beam_1	Parameters
C. Group:	✓ Member type:	
ОК	Save	Column Beam

Pre-defined type of member "simple bar" may be initially opened.

For a chosen member type click the *[Parameters]* button on *Members* tab. It opens MEMBER DEFINITION–PARAMETERS dialog box.

Member Definition - Parameters - SP 16.13330.2011						
Member type: Simple bar		Save				
Buckling (y axis) Member length lef,y:	Buckling (z axis) Member length lef,z: Real	Close				
Ocefficient	Operficient	Service				
Buckling length coefficient Y:	Buckling length coefficient Z:	<u>M</u> ore				
Mi y = 1.00	Mi z = 1.00	Stiffeners				
Section type Y: auto ▼	Section type Z: auto ▼					
Elexural - torsional buckling for tees, chanels and mono-symmetrical I sections.						
Lateral buckling parameters						
Lateral buckling	Lateral buckling length coeff.					
Load level:	lef,b = lo					
Load type:	Lo <u>w</u> er flange lef,b = lo	Help				

Type a new name in the *Member type* editable field. Then change parameters to meet initial data requirements of the current structure. For this particular task for a bar subjected to a combined loads Nc +My+Mz case :

 define buckling parameters - press Buckling length coefficient Y icon which opens BUCKLING DIAGRAMS dialog box and select non-sway structure radio button and the first icon with buckling length coefficient Y equal 1,0;

Buckling types - SP 16.13330.2011 - SP	16.13
$ \begin{array}{c c} \hline \\ \hline \\ 10 \\ \hline \\ 0.7 \\ \hline \\ 0.5 \\ \hline \\ 2.0 \\ \hline \\ 10 \\ \hline 10$	OK Cancel Help

Do the same for the z-z direction.

- switch on *Lateral buckling*
- select required load level by pressing a proper icon



 select appropriate load type by pressing a "load " icon which opens LOAD TYPE dialog box;

Load type	- SP 16.13330.2011	×
	Uniform loads	OK Cancel
-	Concentrated force in center	Help
1/41 ↔	Concentrated force at 1/4 of span le	ength
-	Concentrated force at the end of ca	ntilever
***	Uniform on cantilever	
	Simple bending	

For this task the first icon "uniform loads" was selected.

 define parameters required for member stability verification - press [*More...*] button which opens MEMBER DEFINITION-ADDITIONAL PARAMETERS dialog box and click currently shown "Bending moments for stability analysis" icon ;

Member definition - ad	ditional para	ameters	- SP 16
Bending moments for s	stability analy	sis	
My =	Mz =	4	Cancel
Environment coefficien	nts [Tab. 1]		Help
Section resistance -	gc1 =	1.00	
Member stability -	gc2 =	1.00	
Section factors			
Tension [7.1.1] -	An/A =	1.00	
Shear [8.2.1] -	alpha =	1.00	
Additional conditions fo	or round pipe: ding	s	

In the newly opened MOMENTS FOR CALCULATIONS OF MEMBER STABILITY dialog box select a needed scheme $\ \mbox{icon}$;

Save the newly-created member type under a new name, e.g. "LTB".

MEMBER DEFINITION– PARAMETERS dialog box defined for this verifications looks like:

ember Definition - Parameters	- SP 16.13330.2011	×
Member type: LTB		Save
Buckling (y axis) Member length lef,y: Real Ocefficient	Buckling (z axis) Member length lef,z: Real Coefficient	Close Service
Buckling length coefficient Y: Mi y = 1.00	Buckling length coefficient Z: Mi z = 1.00 $\hat{1.0}$	More
Section type Y: auto Elexural - torsional buckling for mono-symmetrical I sections.	Section type Z: auto ▼ r tees, chanels and	
Lateral buckling parameters		
✓ Lateral buckling	Lateral buckling length coeff.	
Load type:	Lower flange lef,b = lo	Help

In DEFINITIONS dialog box number of the member must be assigned to the appropriate name of *Member type*.

(It is very important when you verify different member types).

Definitions - SP	16.13330.2011	
Members Groups		
N <u>u</u> mber:	1 👻	New
Basic data		
<u>B</u> ar list:	1	
<u>N</u> ame:	beam	Parameters
C. <u>G</u> roup:	✓ Member type:	LTB 👻
ОК	Save	Help

5 Calculations - SP 16.13330.2011 Verification options 1 List Member verification: List Code group verification: List Code group design: Options Optimization Loads Limit state 1 Cases: List Ultimate Calculation archive Serviceability Save calculation results List Configuration OK Calculations Help

In the CALCULATIONS dialog box set for this task :

- Verification option \rightarrow Member Verification
- Loads cases → n° 1
- *Limit state* → only Ultimate Limit state will be analyzed so switch off *Limit stat Serviceability*.

And in CONFIGURATION dialog box opened by [Configuration] button :

- Material factor → gm =1,025; the "1,025" radio button is switched on by default
- Limit slenderness → "Defined by the user" on [Compression] tab - radio button selected in LIMIT SLENDERNESS VALUE dialog box which is opened by [Define] button.

Configuration - SP 16.13330.2011	X
Calculation points	
Number of points: 7	
Characteristic points Options Cancel	
Calculation parameters	
Efficiency ratio: 1.00 Help	
$\boxed{\mathbb{C}} \underbrace{ Components of complex bars are not}_{taken into account}$	
Code parameters	
Structure importance 1.00	
gm=1.025	
Material factor:	
Calculations taking plastic redistribution into consideration	
✓ Limit slenderness: Define	
Compression: Lam,max = 200	
Tension: Lam,max = 250	
Internal forces taken into consideration	
Forces: 📝 Fx 📝 Fy 📝 Fz	
Moments: 📝 Mx 📝 My 📝 Mz	
Units of results	
Camber	
Take deflections from the following case into consideration:	
1 ULS 👻	

it slenderness values - SP 16.13330.2011		
Compression Tension		ОК
STRUCTURE ELEMENTS	LIMIT SLENDERNESS	Cancel
 Chords, cross-braces and support diagonals transmitting support reactions 20 frames, structural and 3D constructions made of pipes and double angles, up to 50 m altitude. 3D frames made of single angles, 3D frames made of pipes and double angles, above 50 m altitude. 	 ○ 180 - 60*alfa ○ 120 	Help
 2. Elements not included in & 1 and 7 a) 2D frames, welded structural and 3D constructions made of single angles, structural and 3D constructions made of pipes and double angles. b) 3D and structural constructions made of single angles with bolted connections. 3. Upper trues chords not fixed during assembly (after completing the assembly assume limit) 	 210 - 60*alfa 220 - 40*alfa 	
slenderness as in & 1) 4. Main columns	© 220	
 Secondary columns, elements of truss columns, elements of vertical stiffeners between columns (under crane girders) 	🔘 210 - 60*alfa	
 Stiffener elements not included in & 5, beams limiting buckling length of compressed elements and other unloaded elements not included in & 7. 	◎ 200	
Compressed and unloaded elements of 3D structures made of T-sections and cross-shaped sections exposed to wind action with the check of slenderness in the vertical	© 150	
8. Limit slenderness defined by the user	200.00	
alpha = N/($Fi^*A^*Ry^*gc2$) and alpha >=0.5		

Now, start verifications by pressing [Calculations] button.

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

1	🗲 SP 16.13330	.2011 - Membe	er Verification	(ULS)1				
	Results Mess	ages						Calc. Note Close
	Member	Section	Material	Lay	Laz	Ratio	Case	Help
	1 beam	<mark>м</mark> ДШ 50х1	C275	37.65	113.03	0.95	1 ULS	
								Analysis Map
								Calculation points Division: none Extremes: none Additional: n = 1
L	1							

<u>Pressing a line with results</u> for the member 1 opens the RESULTS dialog box with detailed results for the analyzed member. The view of the RESULTS windows are presented below.

Simplified results tab

FRESULTS - Code - SP 16	5.13330.2011				X
Дш 50x1	Bar: 1 be	x = 1.00 L = 7. 1 ULS	70 m	Section OK	ОК
Simplified results Detailed	results				Change
INTERNAL FORCES AN N = 800 kN	ND FACTORS My = -400 kN*m mefy = 4.35	Mz = -30 kN*m mefz = 1.12	Qy = 4 kN cy = 1.08	Qz = -244 kN cz = 1.47	
STRESSES IN THE AN SigN = 54.9 MPa	ALYZED SECTION SigMy = -158.9 MPa	SigMz = 66.5 MPa	Ty,max = 0.6 MPa Ty,mid = 0.4 MPa	Tz_max = -51.0 MPa Tz.mid = -45.9 MPa	Forces
	lef,b = 7.70 m Alpha = 5.13	Psi = 2.01 Fi1 = 0.67	,	Fib = 0.67	
BUCKLING y y = 7.70 m lefy = 7.70 m Lamy = 37.65 Lam_y = 1.36	Fiy = 0.91 Ny = 1.50 Fiey = 0.27	₹ <mark>4</mark> 10 BUCKL	ING z lz = 7.70 m lefz = 7.70 m Lamz = 113.03 Lam_z = 4.09	Fiz = 0.44 Fieyz = 0.25 c = 0.49	Calc. Note
SECTION CHECK Web: Lam_w/Lam_uw = (N/(An*Ry*gc1/gn))^1.5(Ty.max / (Rs*gc1/gn)) = (MEMBER STABILITY C Lamy = 37.65 < Lamy.ma My/(Fib*Wcy*Ry*gc2/gn) N/(c*Fiz*A*Ry*gc2/gn)	:0.31 < 1.0; Flange: La 0 + My/(cy*Wyn.min*Ry* 0.00 < 1.00 [8.2.1-(42)]; HECK ax = 200.00 Lamz = 1) = 0.87 < 1.00 [8.4.1-(0.95 < 1.00 [9.2.4-(11	m_f/Lam_uf = 0.42 < 1.0 gc1/gn) + Mz/(cz*Wzn,r Tz.max / (Rs*gc1/gn) : 113.03 < Lamz,max = 2 59)]) [9.4] COMPACT SEC nin*Ry*gc1/gn) = 0.80 < = 0.33 < 1.00 [8.2.1-(4; 00.00 [10.4.1]	CTION 1.00 [9.1.1-(105)] 2)]	

Detailed results tab

ESULIS - Cod	e - SP 16.1333	0.2011				
<mark>тара</mark> (ш. 50х1	Auto	Bar: Load case:	Section x = 1.00 L = 7.70 m 1 ULS	on OK	○○○	ОК
mplified results	Detailed results					Change
Symbol	Value	Unit	Symbol description	Section	^	
Fiz	0.44		Buckling factor	[7.1.3]		
mz	1.21		Relative eccentricity	[9.2.2]		
mefz	1.12		Effective eccentricity	[9.2.2]		Forces
с	0.49		Reduct. factor during stability check	[9.2.5]		
Fieyz	0.25		Reduction factor (compression and bending)	[9.2.9]		
			Verification formulas			
Section chec	k				1	
Section check UFS[LocStab	k 0.42	correct	Web: (hef/t)/Lam,uw; Flange: (bef/tf)/Lam,uf	[9.4]	1	Calc. Note
Section chec UFS[LocStab UFS[Mises]	k 0.42 0.72	correct correct	Web: (hef/t)/Lam,uw; Flange: (bef/tf)/Lam,uf sqrt(Sig^2 + 3.0*Tz,max^2)*0.87/(Ry*gc1/gn)	[9.4] [8.2.1-(44)]		Calc. Note
Section chec UFS[LocStab UFS[Mises] UFS[NMyMzp	k 0.42 0.72 0.80	correct correct	Web: (hef/t)/Lam,uw; Flange: (bef/tf)/Lam,uf sqrt(Sig^2 + 3.0*Tz,max^2)*0.87/(Ry*gc1/gn) (N/(An*Ry*gc1/gn))^1.50 + My/(cy*Wyn,min*Ry*gc1/gn)	[9.4] [8.2.1-(44)] [9.1.1-(105)]		Calc. Note
Section chec UFS[LocStab UFS[Mises] UFS[NMyMzp UFS[Qy]	k 0.42 0.72 0.80 0.00	correct correct correct correct	Web: (hef/t)/Lam,uw; Flange: (bef/tf)/Lam,uf sqrt(Sig^2 + 3.0*Tz,max^2)*0.87/(Ry*gc1/gn) (N/(An*Ry*gc1/gn))^1.50 + My/(cy*Wyn,min*Ry*gc1/gn) Ty,max / (Rs*gc1/gn)	[9.4] [8.2.1-(44)] [9.1.1-(105)] [8.2.1-(42)]	-	Calc. Note
Section chec UFS[LocStab UFS[Mises] UFS[NMyMzp UFS[Qy] UFS[Qz]	k 0.42 0.72 0.80 0.00 0.33	correct correct correct correct correct	Web: (hef/t)/Lam,uw; Flange: (bef/tf)/Lam,uf sqrt(Sig^2 + 3.0*Tz,max^2)*0.87/(Ry*gc1/gn) (N/(An*Ry*gc1/gn))^1.50 + My/(cy*Wyn,min*Ry*gc1/gn) Ty,max / (Rs*gc1/gn) Tz,max / (Rs*gc1/gn)	[9.4] [8.2.1-(44)] [9.1.1-(105)] [8.2.1-(42)] [8.2.1-(42)]	- - - -	Calc. Note
Section chec UFS[LocStab UFS[Mises] UFS[MMyMzp UFS[Qy] UFS[Qz] Member stat	k 0.42 0.72 0.80 0.00 0.33 bility check	correct correct correct correct correct	Web: (hef/t)/Lam,uw; Flange: (bef/tf)/Lam,uf sqrt(Sig^2 + 3.0*Tz,max^2)*0.87/(Ry*gc1/gn) (N/(An*Ry*gc1/gn))^1.50 + My/(cy*Wyn,min*Ry*gc1/gn) Ty,max / (Rs*gc1/gn) Tz,max / (Rs*gc1/gn)	[9.4] [8.2.1-(44)] [9.1.1-(105)] [8.2.1-(42)] [8.2.1-(42)]		C <u>a</u> lc. Note
Section chec UFS[LocStab UFS[Mises] UFS[MMyMzp UFS[Qy] UFS[Qz] Member stat UFB[Lambda]	k 0.42 0.72 0.80 0.00 0.33 0.01 0.33 0.01 0.57	correct correct correct correct correct	Web: (hef/t)/Lam,uw; Flange: (bef/tf)/Lam,uf sqrt(Sig^2 + 3.0*Tz,max^2)*0.87/(Ry*gc1/gn) (N/(An*Ry*gc1/gn))^1.50 + My/(cy*Wyn,min*Ry*gc1/gn) Ty,max / (Rs*gc1/gn) Tz,max / (Rs*gc1/gn) Max (Lamy/Lamy,max ; Lamz/Lamz,max)	[9.4] [8.2.1-(44)] [9.1.1-(105)] [8.2.1-(42)] [8.2.1-(42)] [Table 32]		Calc. Note
Section check UFS[LocStab UFS[Mises] UFS[NMyMzp UFS[Qy] UFS[Qz] UFS[Qz] UFB[Lambda] UFB[Lambda]	k 0.42 0.72 0.80 0.00 0.33 0.1111 (0.657 0.46	correct correct correct correct correct correct correct	Web: (hef/t)/Lam,uw; Flange: (bef/tf)/Lam,uf sqrt(Sig^2 + 3.0*Tz,max^2)*0.87/(Ry*gc1/gn) (N/(An*Ry*gc1/gn))^1.50 + My/(cy*Wyn,min*Ry*gc1/gn) Ty,max / (Rs*gc1/gn) Tz,max / (Rs*gc1/gn) Max (Lamy/Lamy,max ; Lamz/Lamz,max) N/(min(Fiy,Fiz)*A*Ry*gc2/gn)	[9.4] [8.2.1-(44)] [9.1.1-(105)] [8.2.1-(42)] [8.2.1-(42)] [7.able 32] [7.1-(7)]		Calc. Note
Section chect UFS(LocStab UFS(Mises) UFS(NMyMzp UFS(Qy) UFS(Qz) UFS(Qz) UFB(Lambda] UFB(Lambda] UFB(N) UFB(My)	k 0.42 0.72 0.80 0.00 0.33 0ility check 0.57 0.46 0.87	correct correct correct correct correct correct correct correct	Web: (hef/t)/Lam,uw; Flange: (bef/tf)/Lam,uf sqrt(Sig^2 + 3.0*Tz,max^2)*0.87/(Ry*gc1/gn) (N/(An*Ry*gc1/gn))^1.50 + My/(cy*Wyn,min*Ry*gc1/gn) Ty,max / (Rs*gc1/gn) Tz,max / (Rs*gc1/gn) Max (Lamy/Lamy,max ; Lamz/Lamz,max) N/(min(Fiy,Fiz)*A*Ry*gc2/gn)	[9.4] [8.2.1-(44)] [9.1.1-(105)] [8.2.1-(42)] [8.2.1-(42)] [8.2.1-(42)] [7.able 32] [7.1-(7)] [8.4.1-(69)]		Calc. Note
Section chect UFS(LocStab UFS(Mises) UFS(NMyMzp UFS(Qy) UFS(Qz) UFS(Qz) UFB(Lambda] UFB(Lambda] UFB(NMy) UFB(NMy)	k 0.42 0.72 0.80 0.00 0.33 0ility check 0.57 0.46 0.87 0.74	correct correct correct correct correct correct correct correct correct	Web: (hef/t)/Lam,uw; Flange: (bef/tf)/Lam,uf sqrt(Sig*2 + 3.0*Tz,max*2)*0.87/(Ry*gc1/gn) (N/(An*Ry*gc1/gn))^1.50 + My/(cy*Wyn,min*Ry*gc1/gn) Ty,max / (Rs*gc1/gn) Tz,max / (Rs*gc1/gn) Max (Lamy/Lamy,max ; Lamz/Lamz,max) N/(min(Fiy,Fiz)*A*Ry*gc2/gn) My/(Fib*Wcy*Ry*gc2/gn) N/(Fiey*A*Ry*gc2/gn)	[9.4] [8.2.1-(44)] [9.1.1-(105)] [8.2.1-(42)] [8.2.1-(42)] [8.2.1-(42)] [7.1-(42)] [7.1-(7)] [8.4.1-(69)] [9.2.2-(109)]		Calc. Note
Section chect UFS[LocStab UFS[Mises] UFS[MMyMzp UFS[Qy] UFS[Qz] Member stat UFB[Lambda] UFB[Lambda] UFB[My] UFB[MMy] UFB[MMy]	k 0.42 0.72 0.80 0.00 0.33 0ility check 0.57 0.46 0.87 0.74 0.95	correct correct correct correct correct correct correct correct correct correct	Web: (hef/t)/Lam,uw; Flange: (bef/tf)/Lam,uf sqrt(Sig*2 + 3.0*Tz,max*2)*0.87/(Ry*gc1/gn) (N/(An*Ry*gc1/gn))*1.50 + My/(cy*Wyn,min*Ry*gc1/gn) Ty,max / (Rs*gc1/gn) Tz,max / (Rs*gc1/gn) Max (Lamy/Lamy,max ; Lamz/Lamz,max) N/(min(Fiy,Fiz)*A*Ry*gc2/gn) My/(Fib*Wcy*Ry*gc2/gn) N/(reir[z*A*Ry*gc2/gn) N/(c*Fiz*A*Ry*gc2/gn)	[9.4] [8.2.1-(44)] [9.1.1-(105)] [8.2.1-(42)] [8.2.1-(42)] [7.1-(42)] [7.1-(7)] [8.4.1-(69)] [9.2.2-(109)] [9.2.4-(111)]		Calc. Note

Pressing the [*Calc.Note*] button in "RESULTS –Code" dialog box 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.

	ST	EEL DESIGN	
CODE: SP 16.13330 ANALYSIS TYPE: M	2011* Steel structures. Iember Verification		
CODE GROUP: MEMBER: 1 beam	POINT:	COORDINATE	E: $x = 1.00 L = 7.70 m$
LOADS:			
Governing Load Case:	1 ULS		
MATERIAL C275			
Ry = 270.0 MPa	Rs = 156.6 MPa	Ru = 370.0 MPa	E = 206000.0 MPa
gu = 1.30	gc1 = 1.00	gc2 = 1.00	gn = 1.00
	ARAMETERS: ДШ 50x1		
ht=48.4 cm bf=30.0 cm	Av=90.00 cm2	Az=53.24 cm2	Ax=145 70 cm2
t=1.1 cm	Iy=60930.00 cm4	Iz=6762.00 cm4	Ix=88.97 cm4
tf=1.5 cm	Wely=2517.77 cm3	Welz=450.80 cm3	•
Web:	hef/t = 36.55	Lam,uw = 117.87	
Flange:	bef/tf = 7.90	Lam,uf = 19.03	
INTERNAL FORCES	AND FACTORS		
N = 800 kN	Mv = -400 kN*m	Mz = -30 kN*m	
	mefv = 4.35	mefz = 1.12	
	$Q_V = 4 \text{ kN}$	Qz = -244 kN	
	cy = 1.08	cz = 1.47	
STRESSES AT CHAI	RACTERISTIC SECTION F		
SigN = 54.9 MPa	SigMy = -158.9 MPa	SigMz = 66.5 MPa	
-			
	Ty,max = 0.6 MPa	$Tz_max = -51.0 MPa$	
	Ty,mid = 0.4 MPa	Tz,mid = -45.9 MPa	
Left $= 7.70 \text{m}$	L BUCKLING PARAMETE	:R5:	
Alpha = 5.13	Fi1 = 0.67	Fib = 0.67	
	TEDC.		
	avie:	tu 👫 About 7 avi	e.
1v = 7.70 m	Fiv = 0.91	1z = 7.70 m	Fiz = 0.44
1efy = 7.70 m	Ny = 1.50	1efz = 7.70 m	
Lamy = 37.65	Fiey = 0.27	Lamz = 113.03	Fieyz = 0.25
Lam_y = 1.36	-	Lam_z = 4.09	c = 0.49
VERIFICATION FOR <u>Section check</u> Web: Lam_w/Lam_uw= (N/(An*Ry*gc1/gn))^1 Ty,max / (Rs*gc1/gn) = <u>Member stability check</u>	MULAS: = 0.31 < 1.0; Flange: Lam_f/ .50 + My/(cy*Wyn,min*Ry* 0.00 < 1.00 [8.2.1-(42)]; Tz	Lam_uf = 0.42 < 1.0 [9.4] CON gc1/gn) + Mz/(cz*Wzn,min*Ry ,max / (Rs*gc1/gn) = 0.33 < 1.00	MPACT SECTION *gc1/gn) = 0.80 < 1.0 [8.2.1-(42)]
Lamy = 37.65 < Lamy,ma	ax = 200.00 Lamz = 113.0	3 < Lamz,max = 200.00 [10.4.1]	
My/(Fib*Wey*Ry*ge?)	$(\sigma n) = 0.87 < 1.00$ [8.4.1_(69)	1	

 $\begin{array}{l} My/(Fib*Wcy*Ry*gc2/gn) = 0.87 < 1.00 \quad [8.4.1-(69)] \\ N/(c*Fiz*A*Ry*gc2/gn) = 0.95 < 1.00 \quad [9.2.4-(111)] \end{array}$

Section OK !!!