Autodesk®

Robot™ Structural Analysis Professional

VERIFICATION MANUAL

FOR STEEL MEMBERS DESIGN

March 2014

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:

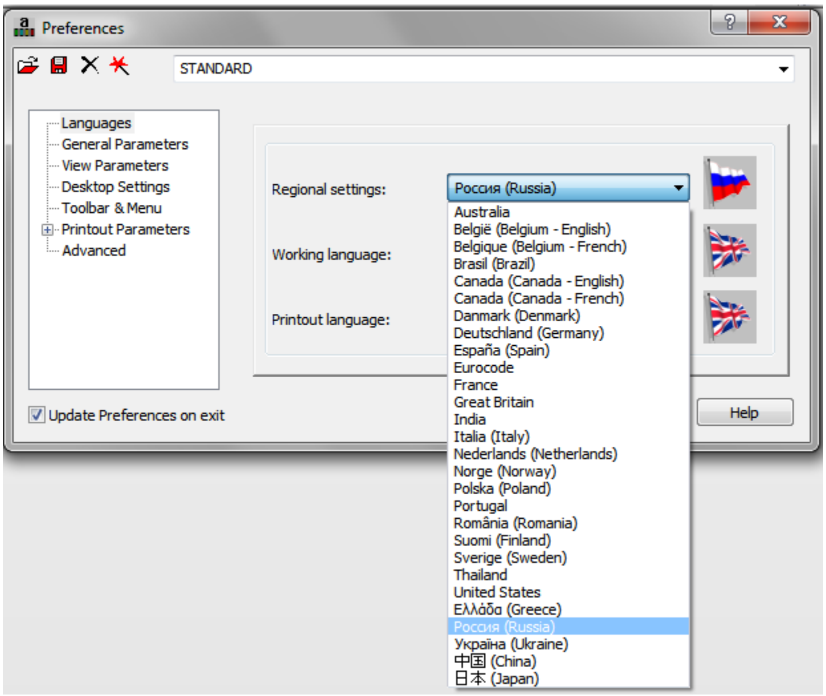
- title of the problem
- specification of the problem
- Robot solution of the problem
- outputs with calculation results and calculation notes

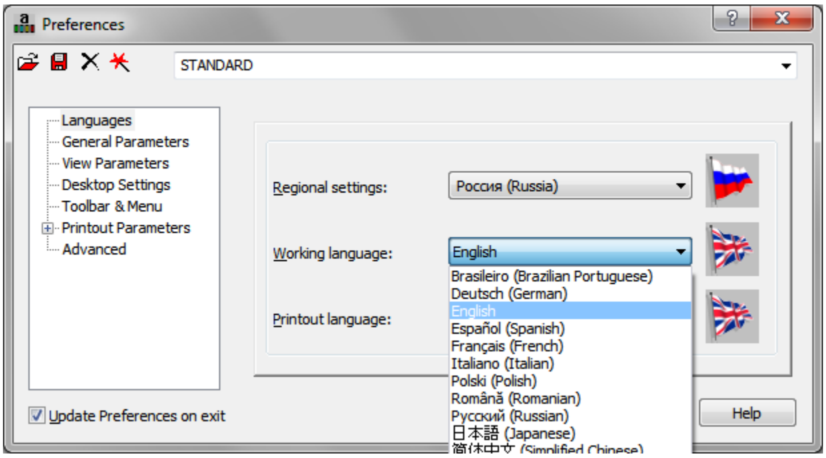
GENERAL REMARKS

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

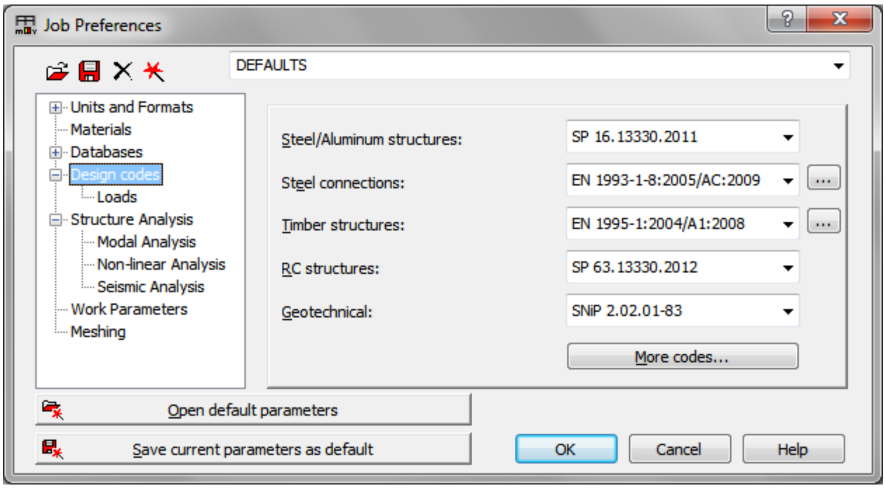
To specify required **regional preferences** click Tools / [Preferences...] and in default opened *Preferences* dialog box select in combo boxes a needed country (region) and working / printout language.



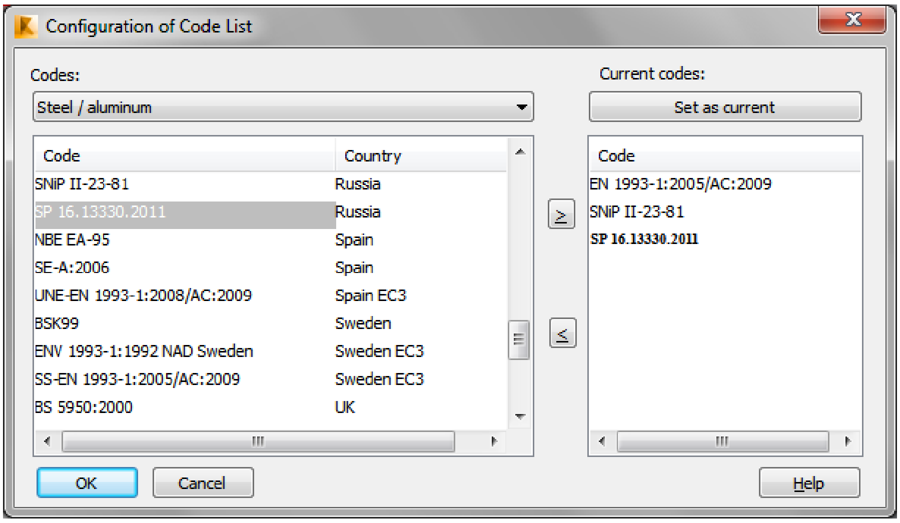


B. Job Preferences

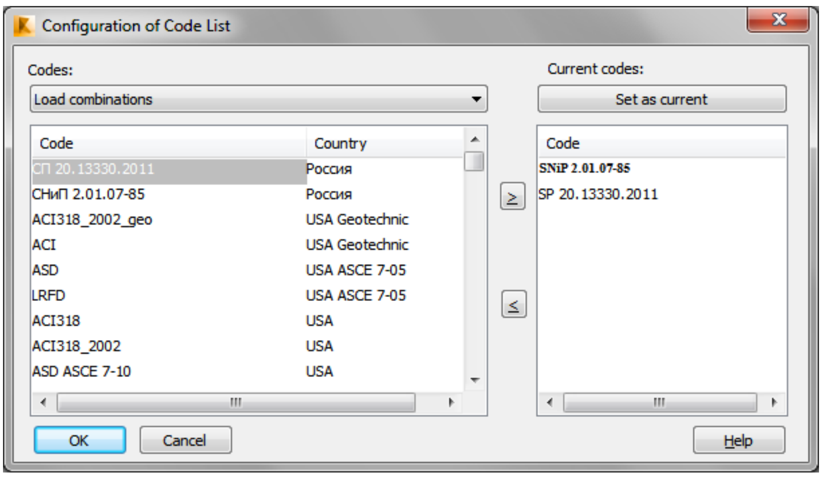
To specify required **job preferences** click Tools / [Job Preferences...] and in default opened *Job Preferences* dialog box select preferences corresponding to your structure example at the left list view and appropriate combo boxes. Below there is a screenshot showing the selection [Design codes]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



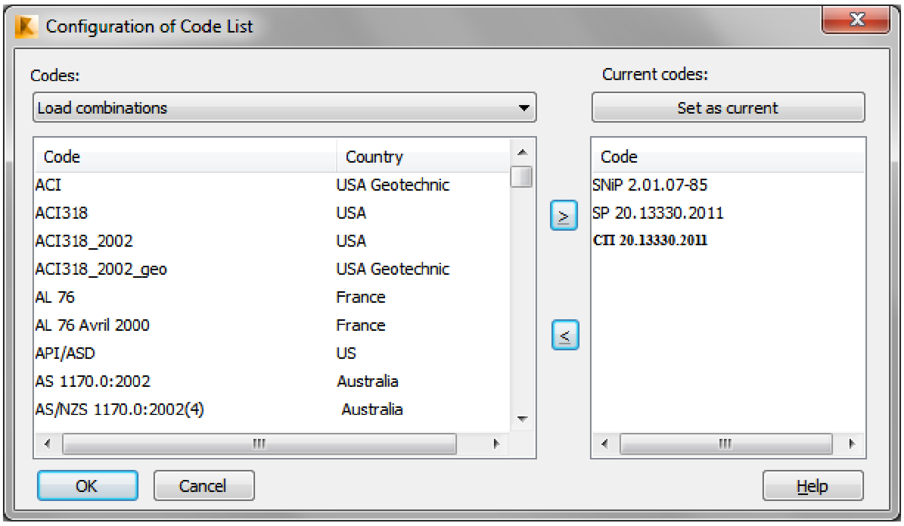
You can create a new Job Preferences with arbitrarily chosen options and parameters (standards, materials, databases, load codes etc.) appropriate for required condition under a new name to make it easier for future work .
In that case choose proper detailed data from [*Configuration of Code List]* dialog box which is opened after pressing [*More codes…]* button and e.g. looks as shown below :



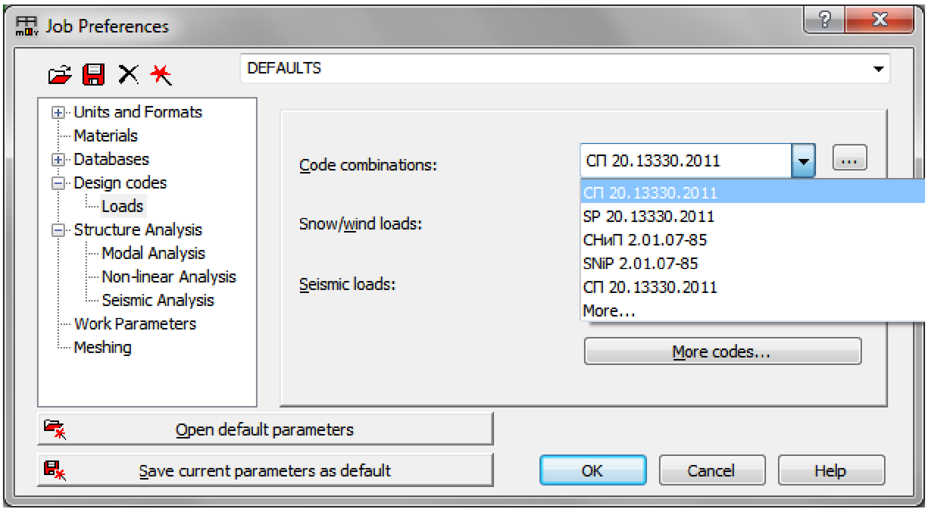
If you pick *Load combinations* from *Codes* combo box the new list view appears:



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



Press OK.



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

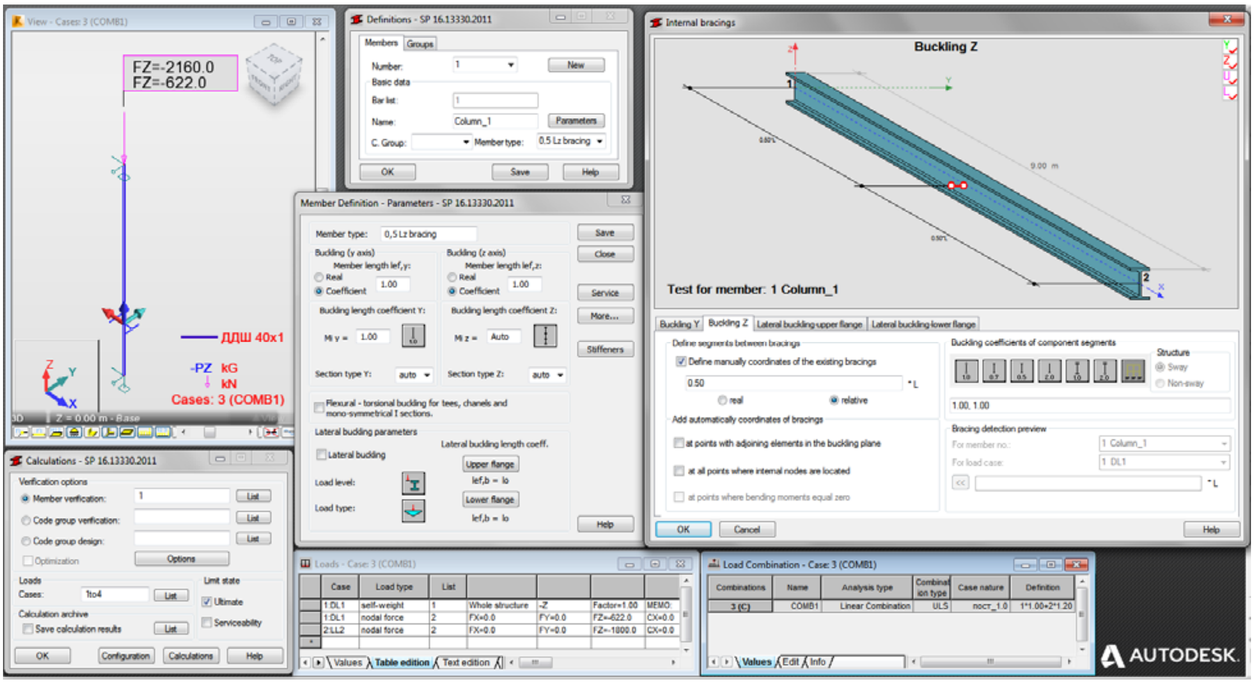
File: SP16_ex_Compression.rtd

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 .

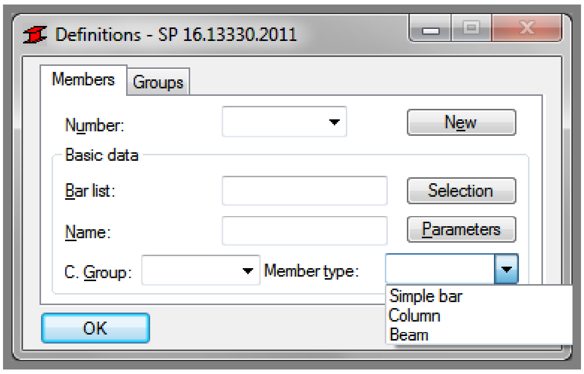


SOLUTION:

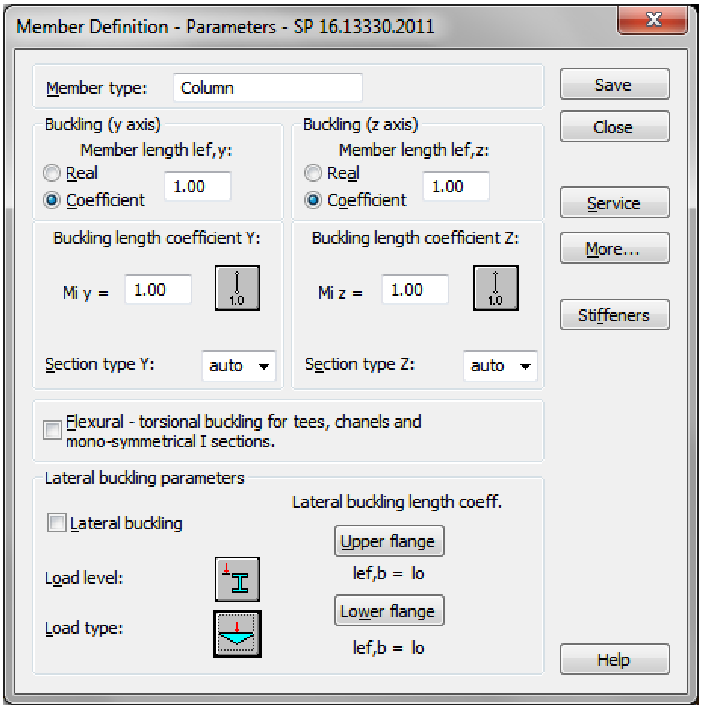
You must remember to specify appropriate gm material factor (a new option) in CONFIGURATION dialog box placed at Steel/Aluminium Design level.

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.

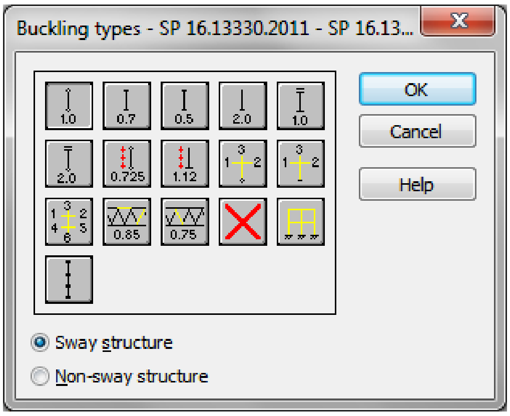
Pre-defined type of member "*Column*" may be initially opened.



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

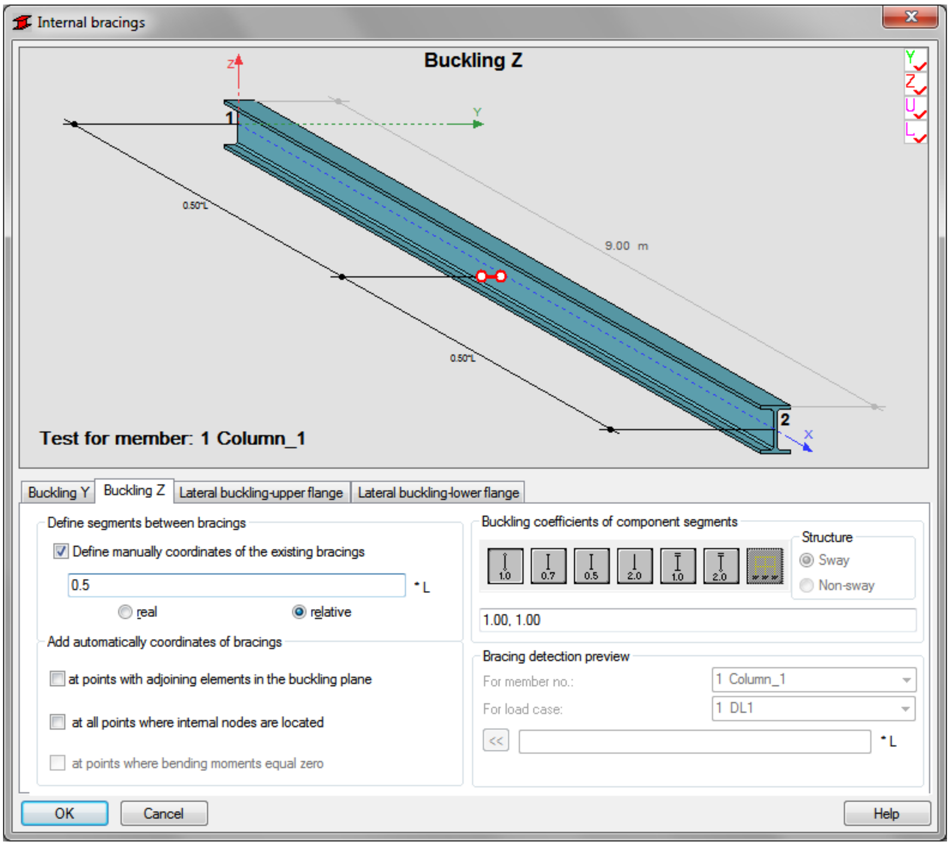


Type a new name in the *Member type* editable field. Change parameters to meet initial data requirements of a structure in a current task. In this particular compression case define buckling z-z parameters. Press *Buckling length coefficient Z* icon which opens BUCKLING DIAGRAMS dialog box.



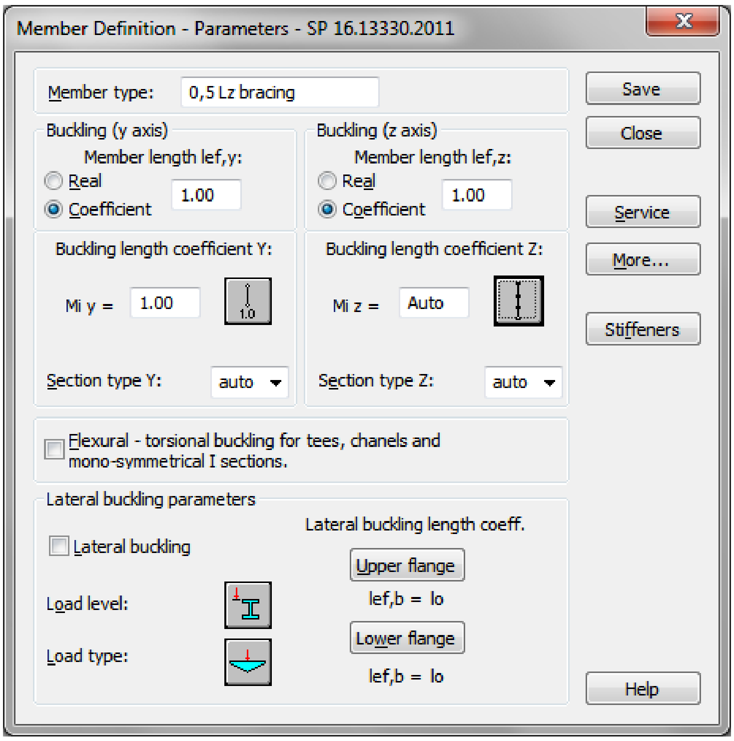
Click a last icon.

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.



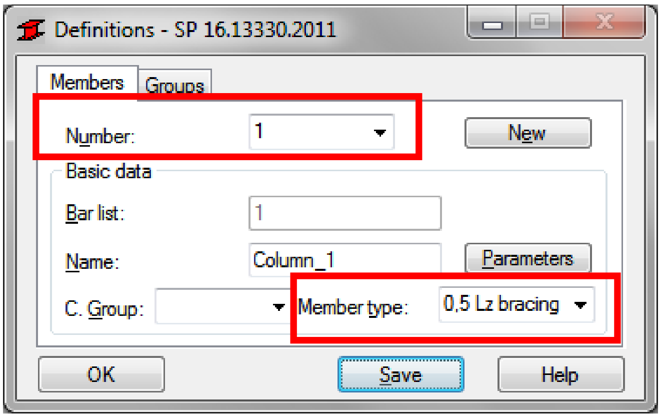
Press OK.

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

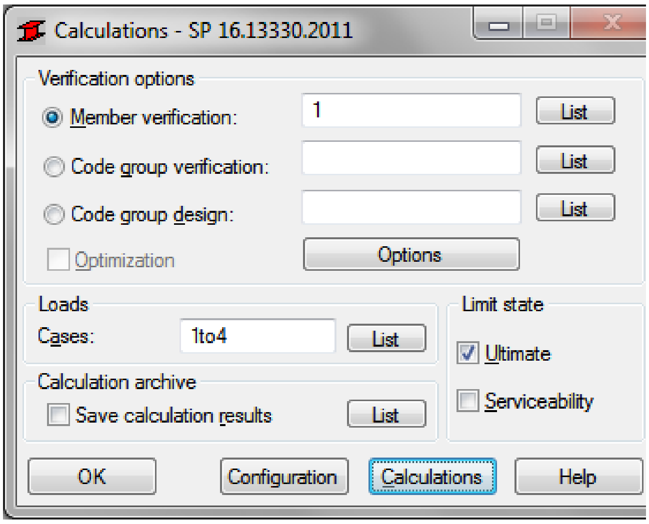


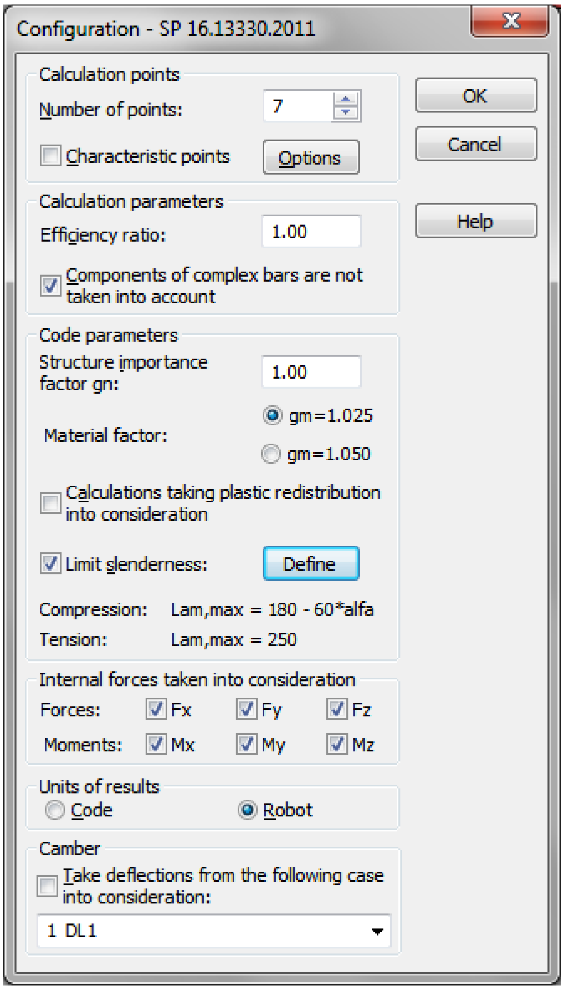
Number of the member must be assigned to appropriate name of *Member type.*

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

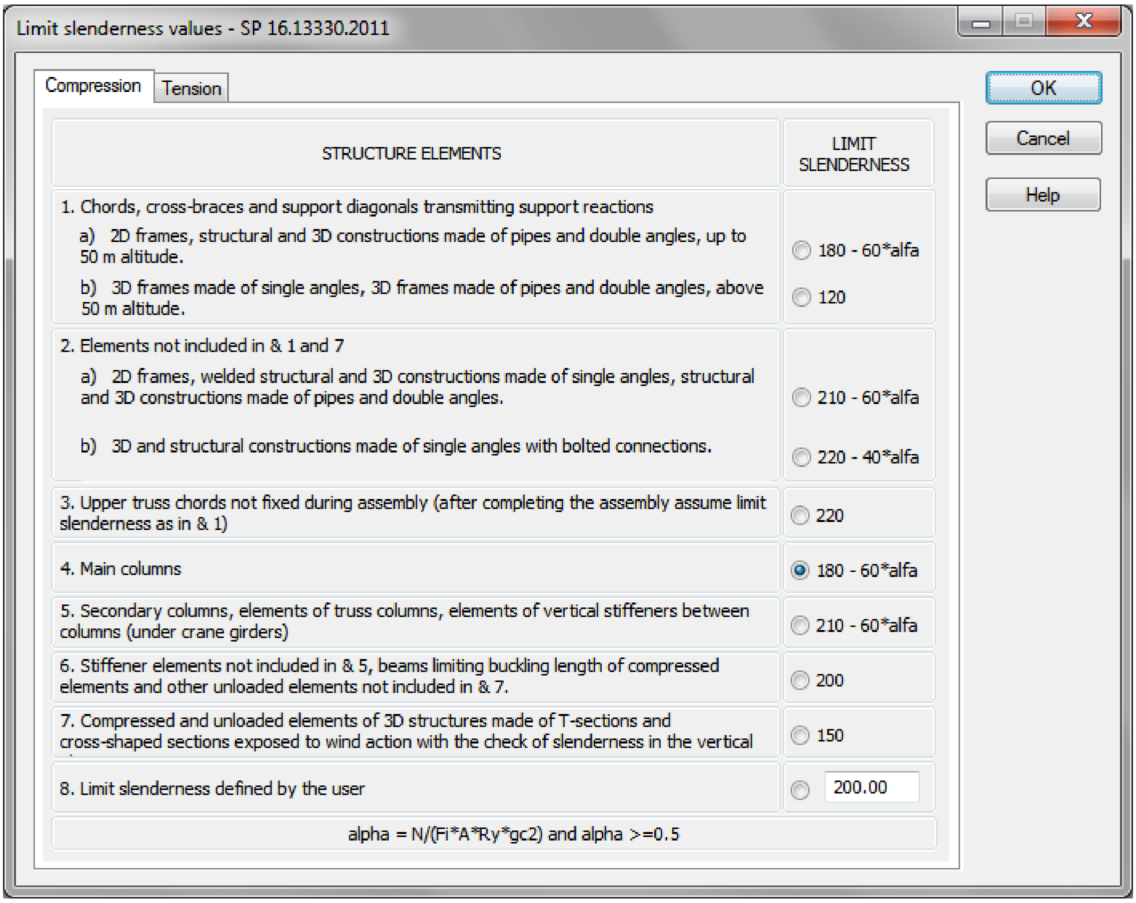


In the CALCULATIONS dialog box set for this task:



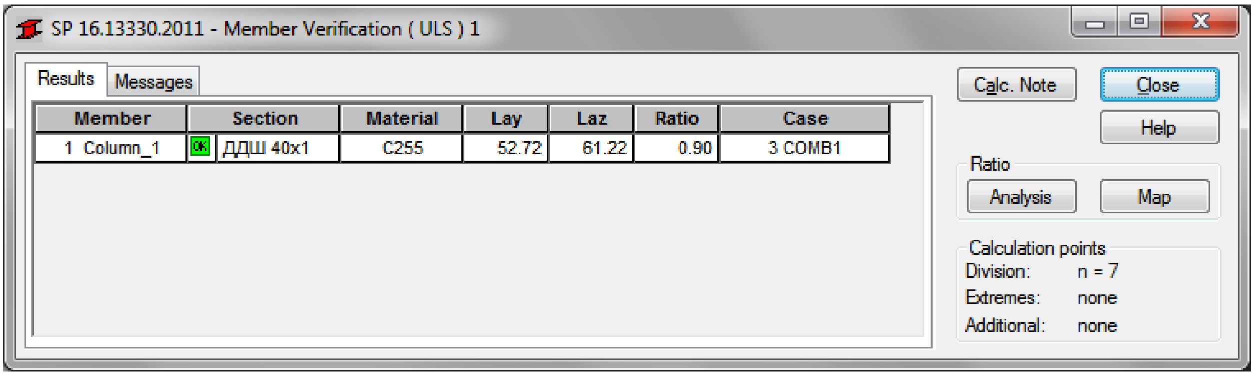


- *Verification option* → Member Verification
- *Loads cases* → 1 to 4
- *Limit state* → only Ultimate Limit state will be analyzed so switch off *Limit stat Serviceability*
- *Material factor* → *gm =1,025 ;* *the "1,025" radio button* is switched on by default in CONFIGURATION dialog box (opened by [Configuration] button)
- *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



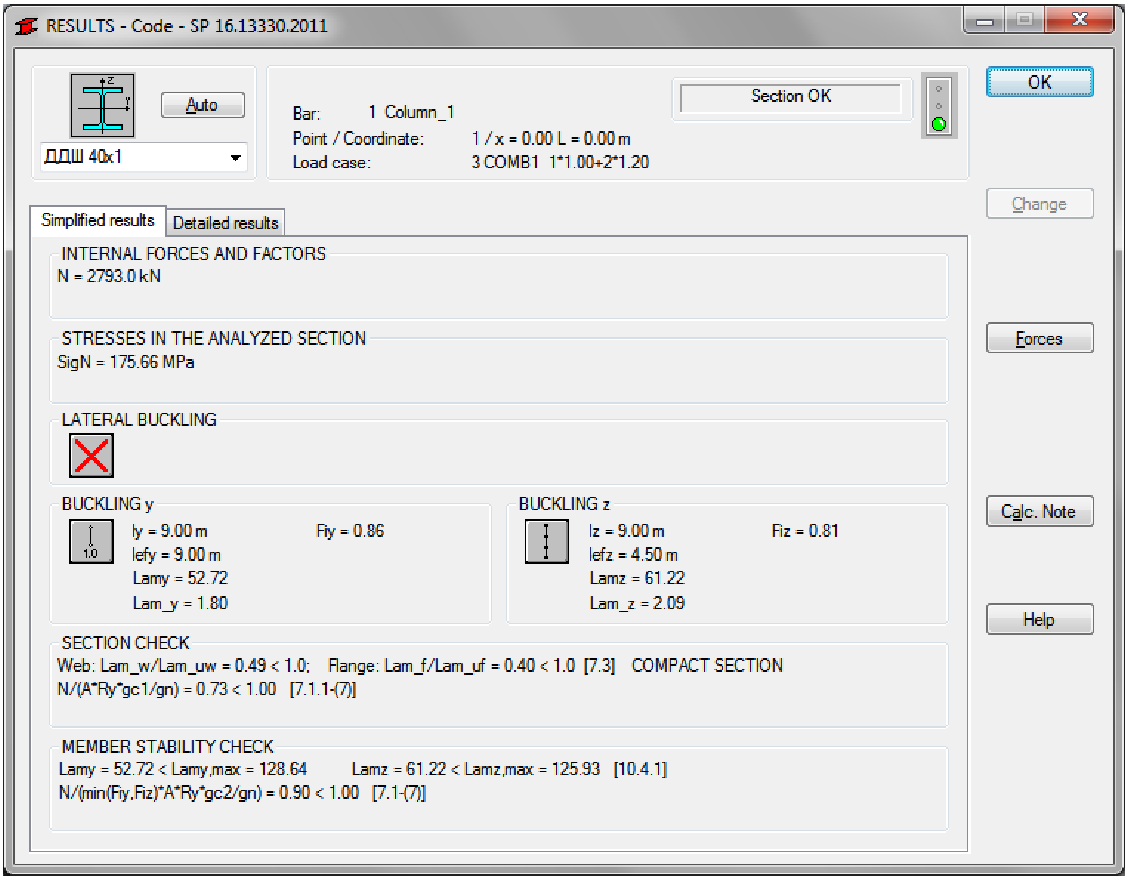
Now, start calculations by pressing *Calculations* button.

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

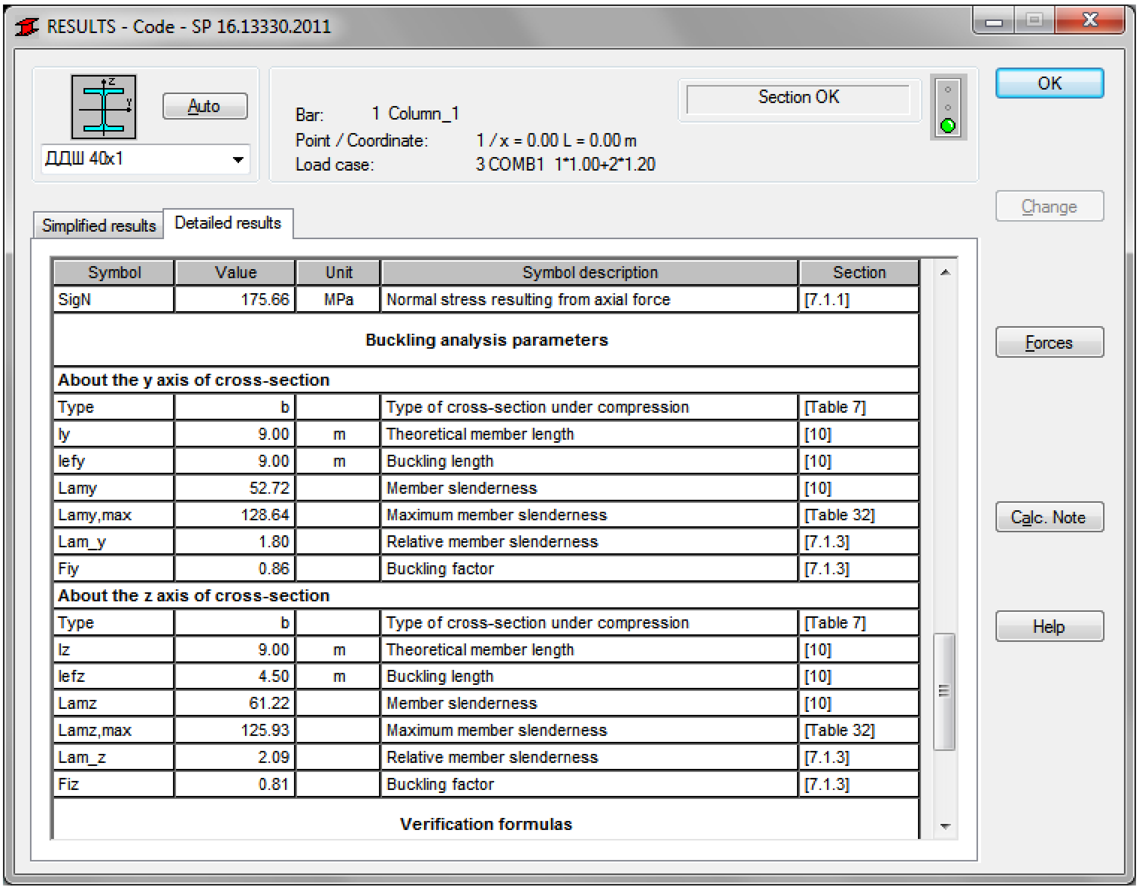


Pressing the line with results for the member 1 opens the RESULTS dialog box with detailed results for the analyzed member. The views of the RESULTS windows are presented below.

Simplified results tab



Detailed results tab



Symbol	Value	Unit	Symbol description	Section
SigN	175.66	MPa	Normal stress resulting from axial force	[7.1.1]
Buckling analysis parameters				
About the y axis of cross-section				
Type	b		Type of cross-section under compression	[Table 7]
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]
Lam_y	1.80		Relative member slenderness	[7.1.3]
Fiy	0.86		Buckling factor	[7.1.3]
About the z axis of cross-section				
Type	b		Type of cross-section under compression	[Table 7]
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.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.

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

STEEL DESIGN

CODE: *SP 16.13330.2011* Steel structures.*
ANALYSIS TYPE: Member Verification

CODE GROUP:
MEMBER: 1 Column_1 **POINT:** 1 **COORDINATE:** 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	Rs = 139.20 MPa	Ru = 360.00 MPa	E = 206000.00 MPa
gu = 1.30	gc1 = 1.00	gc2 = 1.00	gn = 1.00

SECTION PARAMETERS: **ДДШ 40x1**

ht=39.8 cm			
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 Lam,uw = 56.58
Flange: bef/tf = 6.59 Lam,uf = 16.67

INTERNAL FORCES AND FACTORS
N = 2793.0 kN

STRESSES AT CHARACTERISTIC SECTION POINTS
SigN = 175.66 MPa

LATERAL BUCKLING PARAMETERS:

BUCKLING PARAMETERS:

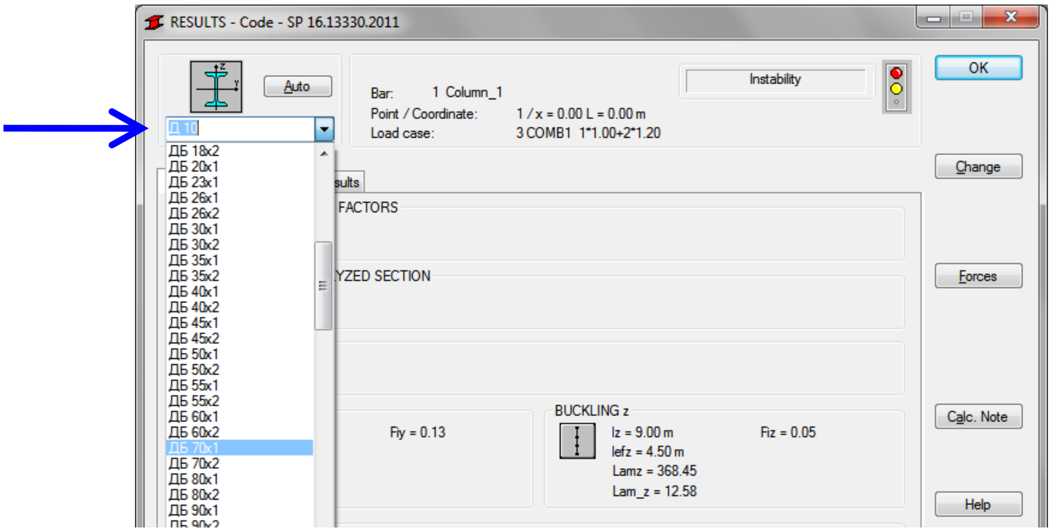
About Y axis:		About Z axis:	
ly = 9.00 m	Fiy = 0.86	lz = 9.00 m	Fiz = 0.81
lefy = 9.00 m		lefz = 4.50 m	
Lamy = 52.72		Lamz = 61.22	
Lam_y = 1.80		Lam_z = 2.09	

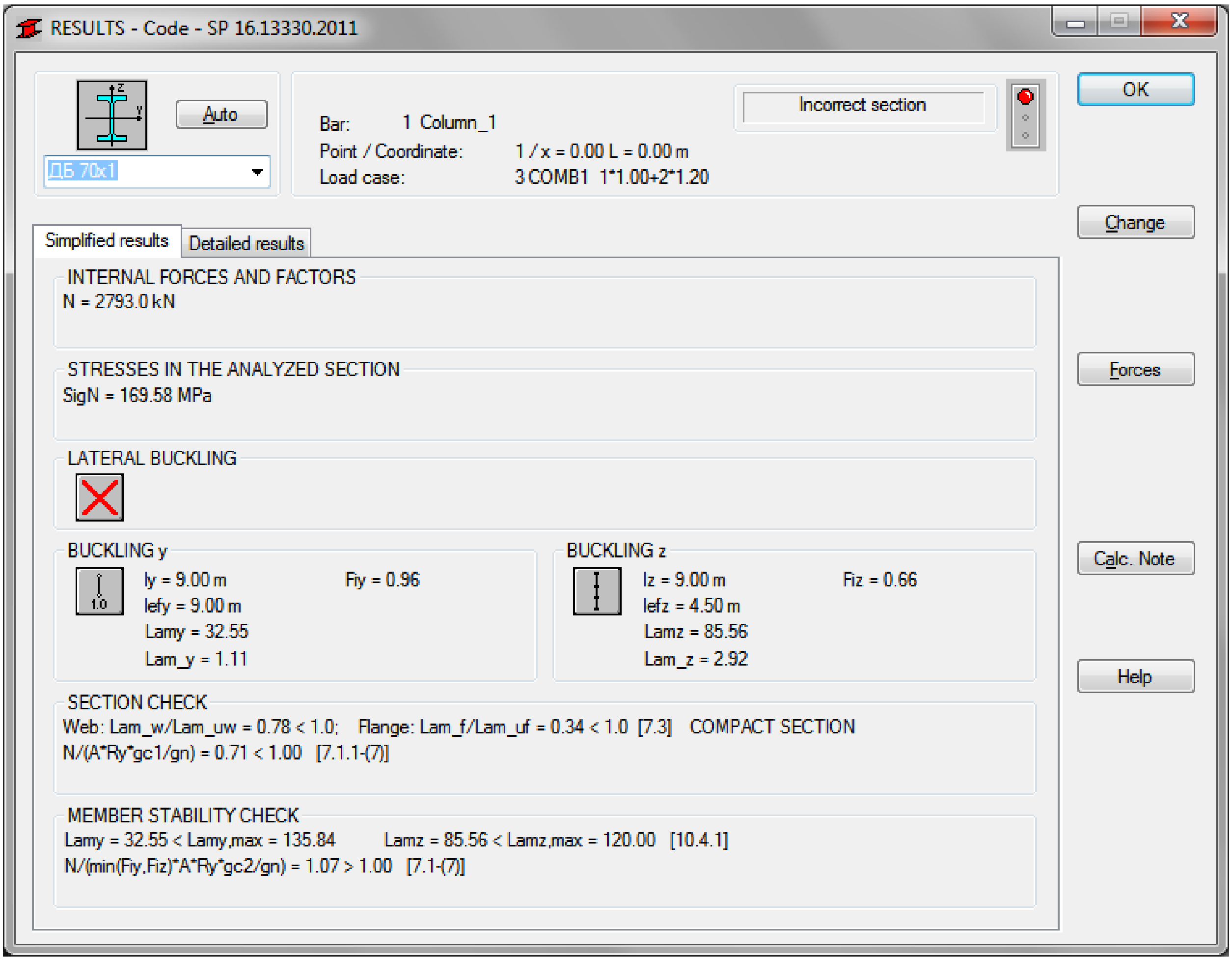
VERIFICATION FORMULAS:
Section check
Web: Lam_w/Lam_uw = 0.49 < 1.0; Flange: Lam_f/Lam_uf = 0.40 < 1.0 [7.3] COMPACT SECTION
N/(A*Ry*gc1/gn) = 0.73 < 1.00 [7.1.1-(5)]

Member stability check
Lamy = 52.72 < Lamy,max = 128.64 Lamz = 61.22 < Lamz,max = 125.93 [10.4.1]
N/(min(Fiy,Fiz)*A*Ry*gc2/gn) = 0.90 < 1.00 [7.1-(7)]

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.





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

STEEL DESIGN

CODE: *SP 16.13330.2011* Steel structures.*
ANALYSIS TYPE: Member Verification

CODE GROUP:
MEMBER: 1 Column_1 **POINT:** 1 **COORDINATE:** 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	Rs = 139.20 MPa	Ru = 360.00 MPa	E = 206000.00 MPa
gu = 1.30	gc1 = 1.00	gc2 = 1.00	gn = 1.00

SECTION PARAMETERS: ДБ 70x1

ht=69.1 cm			
bf=26.0 cm	Ay=80.60 cm2	Az=82.92 cm2	Ax=164.70 cm2
t=1.2 cm	Iy=125930.01 cm4	Iz=4556.00 cm4	Ix=104.35 cm4
tf=1.6 cm	Wely=3644.86 cm3	Welz=350.46 cm3	
Web:	hef/t = 51.00	Lam,uw = 65.10	
Flange:	bef/tf = 6.45	Lam,uf = 19.10	

INTERNAL FORCES AND FACTORS
N = 2793.0 kN

STRESSES AT CHARACTERISTIC SECTION POINTS
SigN = 169.58 MPa

LATERAL BUCKLING PARAMETERS:

BUCKLING PARAMETERS:

About Y axis:		About Z axis:	
ly = 9.00 m	Fiy = 0.96	lz = 9.00 m	Fiz = 0.66
lefy = 9.00 m		lefz = 4.50 m	
Lamy = 32.55		Lamz = 85.56	
Lam_y = 1.11		Lam_z = 2.92	

VERIFICATION FORMULAS:
Section check
Web: Lam_w/Lam_uw = 0.78 < 1.0; Flange: Lam_f/Lam_uf = 0.34 < 1.0 [7.3] COMPACT SECTION
N/(A*Ry*gc1/gn) = 0.71 < 1.00 [7.1.1-(7)]
Member stability check
Lamy = 32.55 < Lamy,max = 135.84 Lamz = 85.56 < Lamz,max = 120.00 [10.4.1]
N/(min(Fiy,Fiz)*A*Ry*gc2/gn) = 1.07 > 1.00 [7.1-(7)]

Incorrect section !!!

VERIFICATION PROBLEM 2
Lateral-torsional buckling of beams

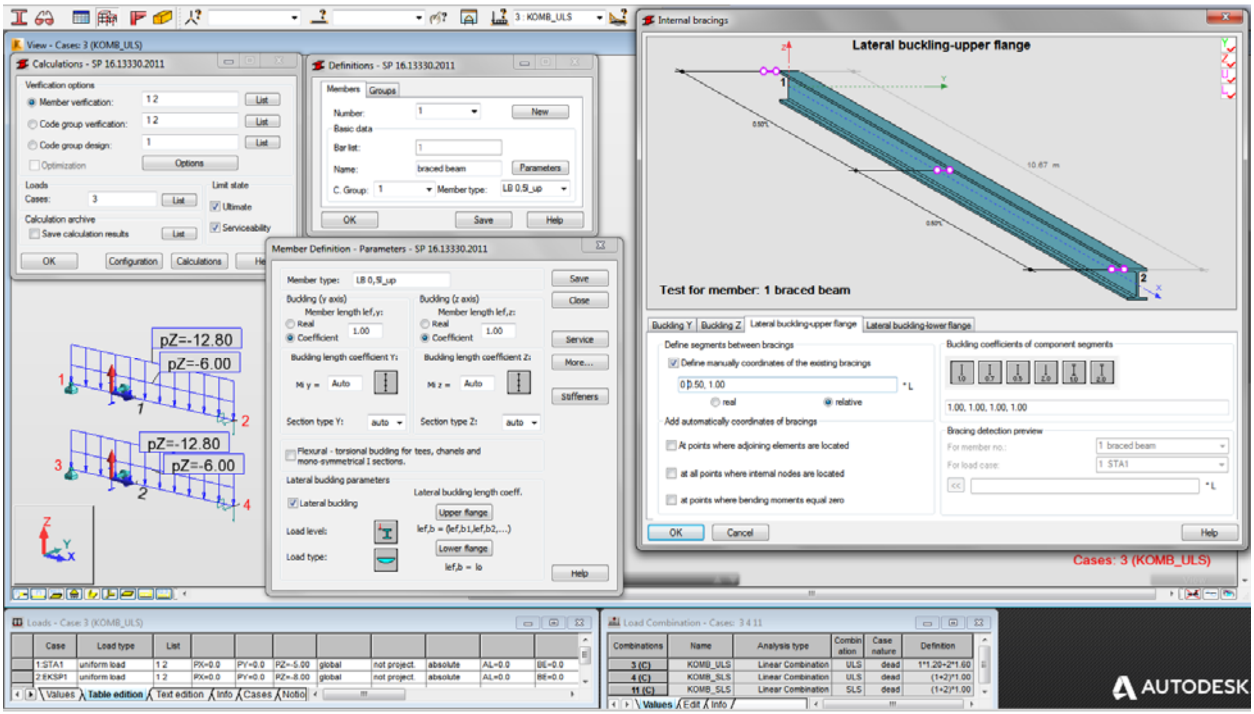
File: SP16_ex_LTB beam.rtd

TITLE:

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

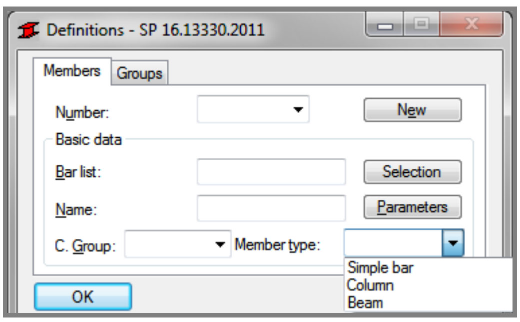
SPECIFICATION:

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.



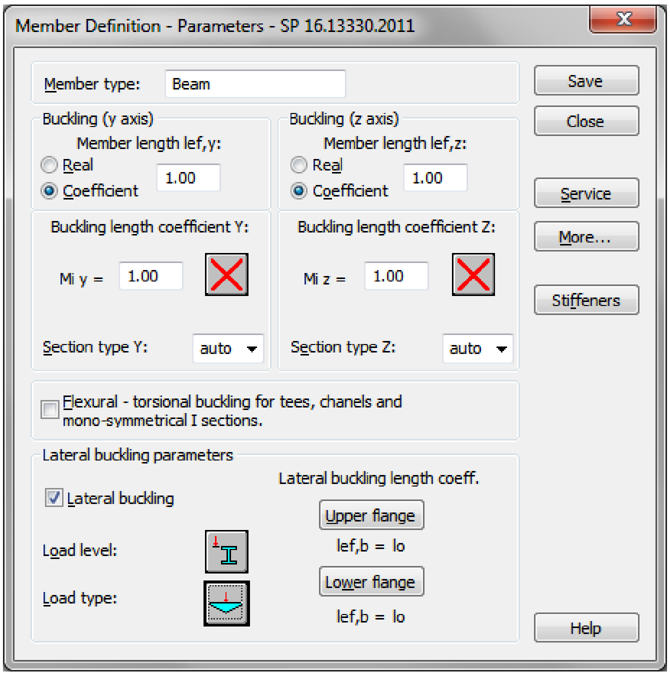
SOLUTION:

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.



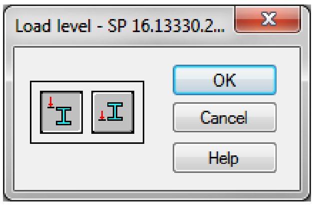
Pre-defined "*beam*" member type may be initially opened.

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

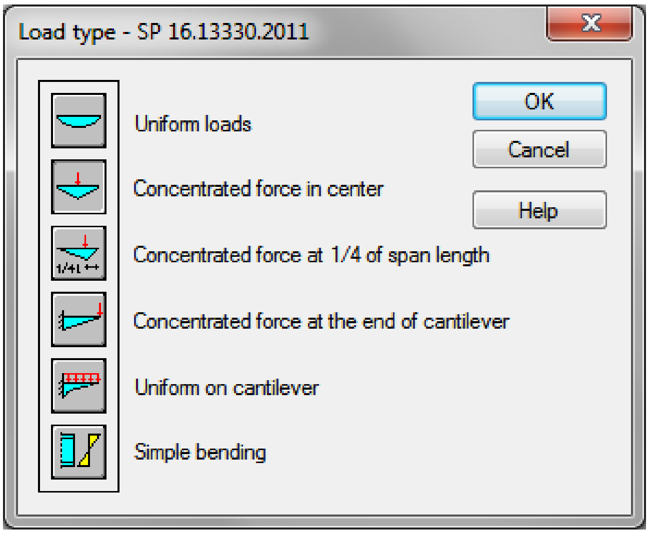


Type a new name in the *Member type* editable field. Then, change parameters to meet initial data requirements of the structure. In this particular bending case set the following lateral-buckling parameters :

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



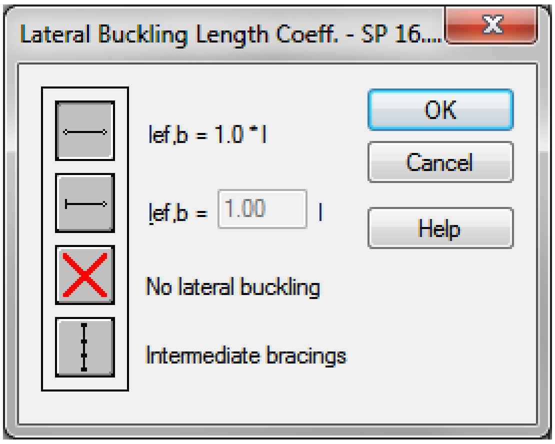
- define appropriate load type by pressing “ load “ icon which opens LOAD TYPE dialog box;



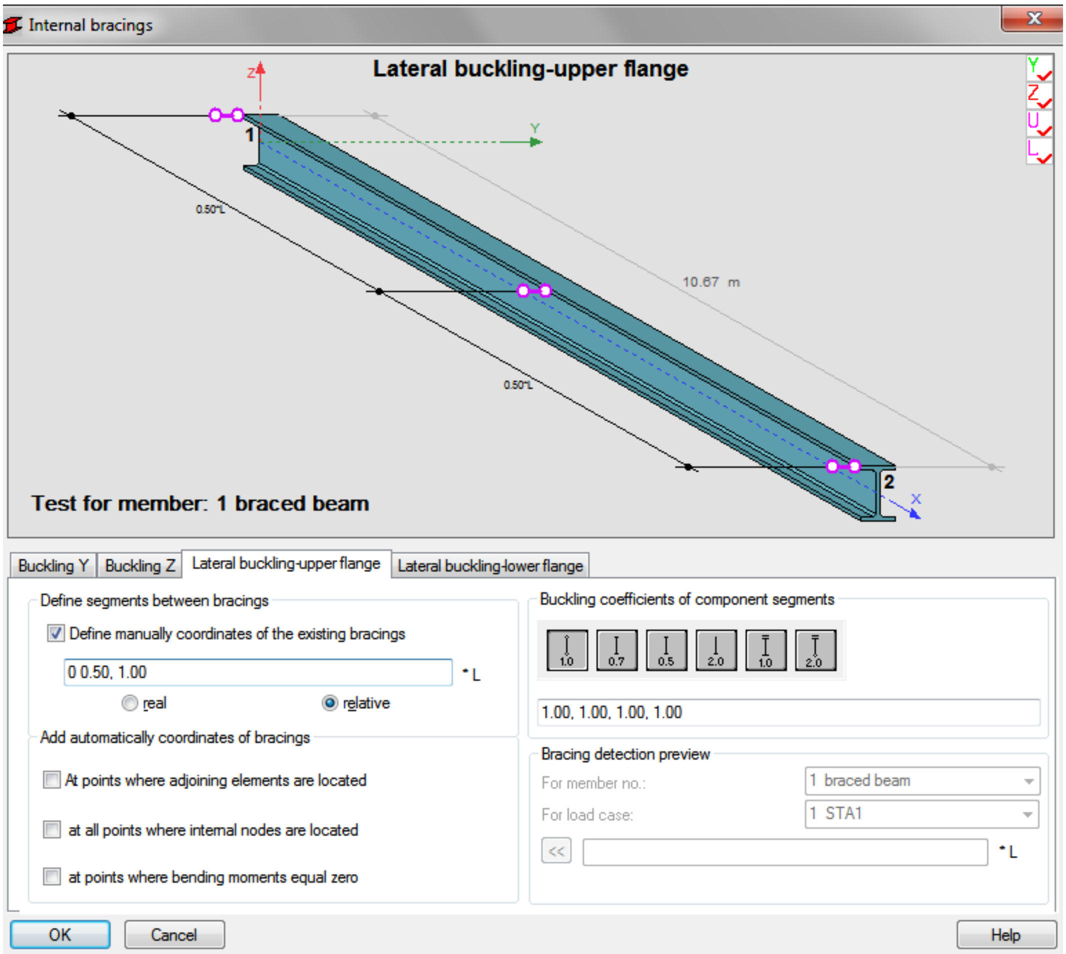
for this task the first icon “uniform loads“ was selected.

- 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.

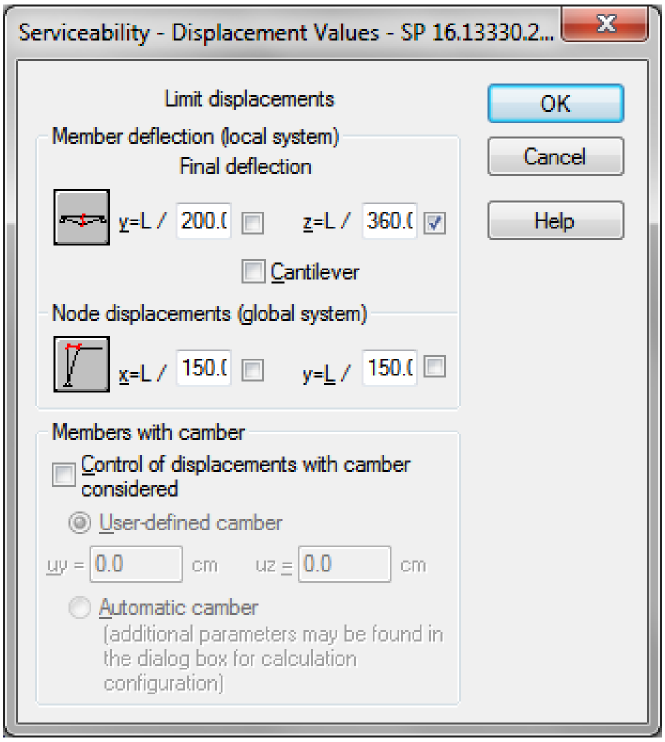


Click the last icon *Intermediate bracings*. The new dialog box *INTERNAL BRACINGS* will appear with automatically active *Lateral buckling - Upper flange* tab .
In *INTERNAL BRACINGS* dialog box there are possibilities of defining independent bracings for buckling and lateral buckling of the marked *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.

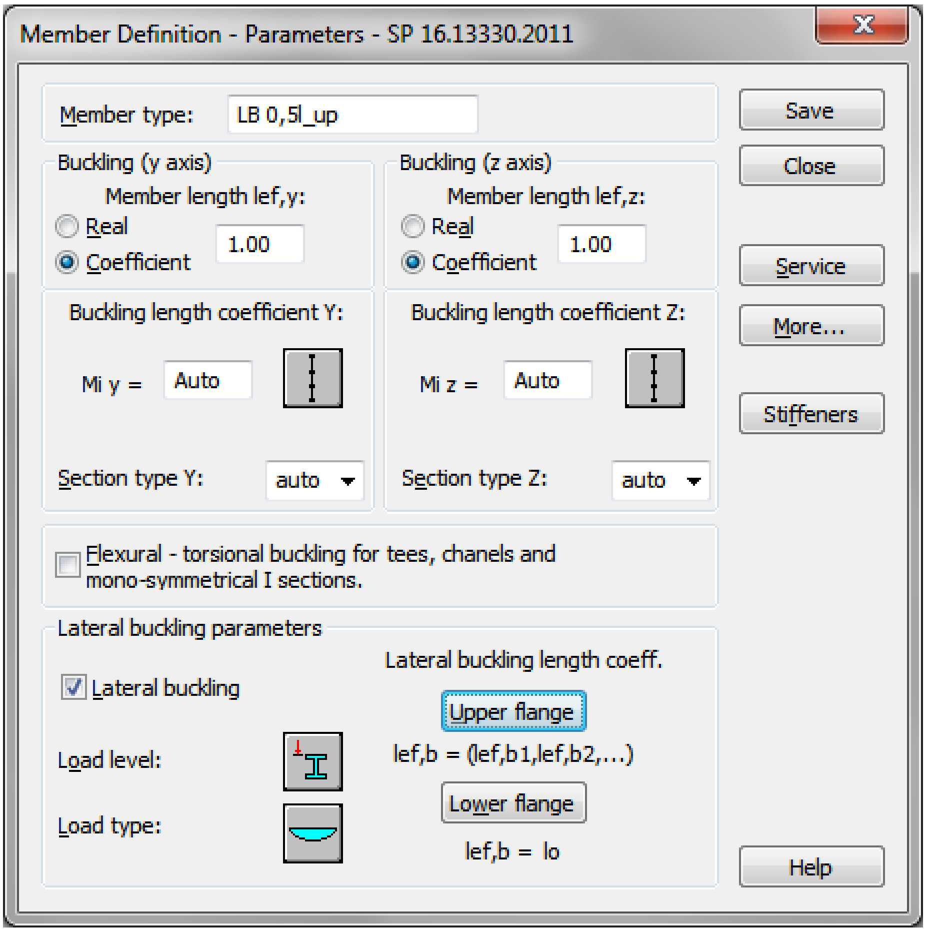


Press OK.

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

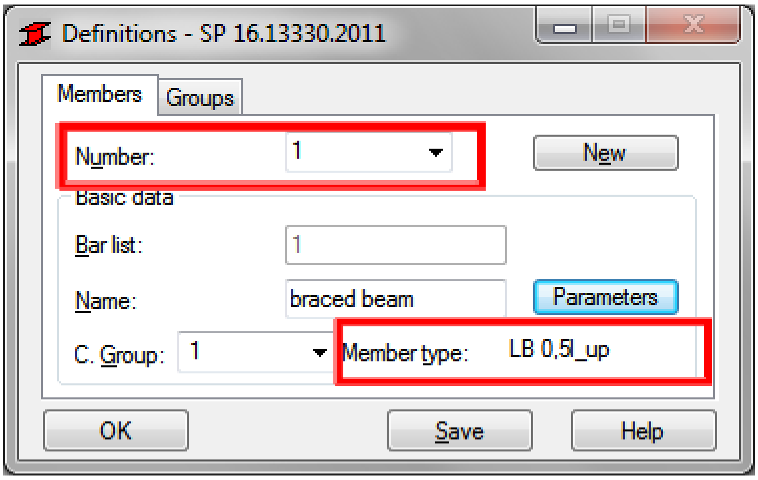


Save the newly-created member type under a new name , e.g. as "LB 0,5l up"

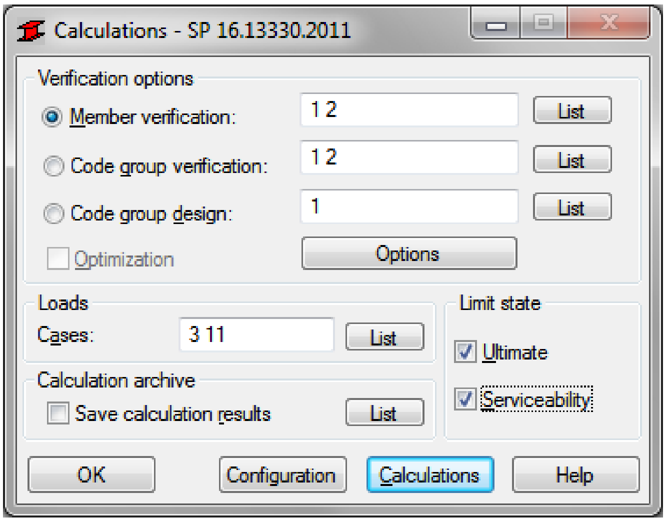


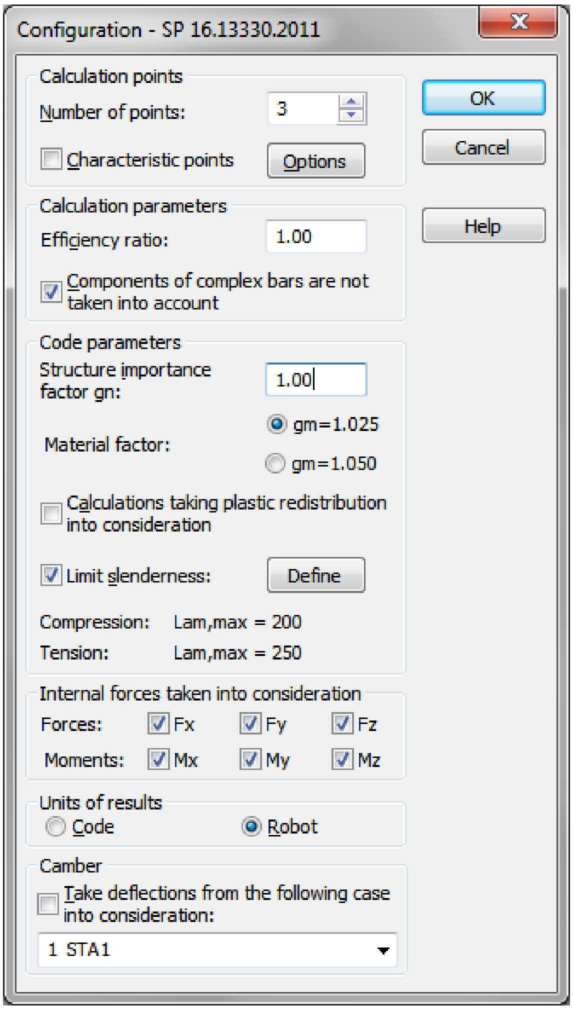
Number of a member must be assigned to the appropriate name of *Member type.*

It is very important when you verify different member types.

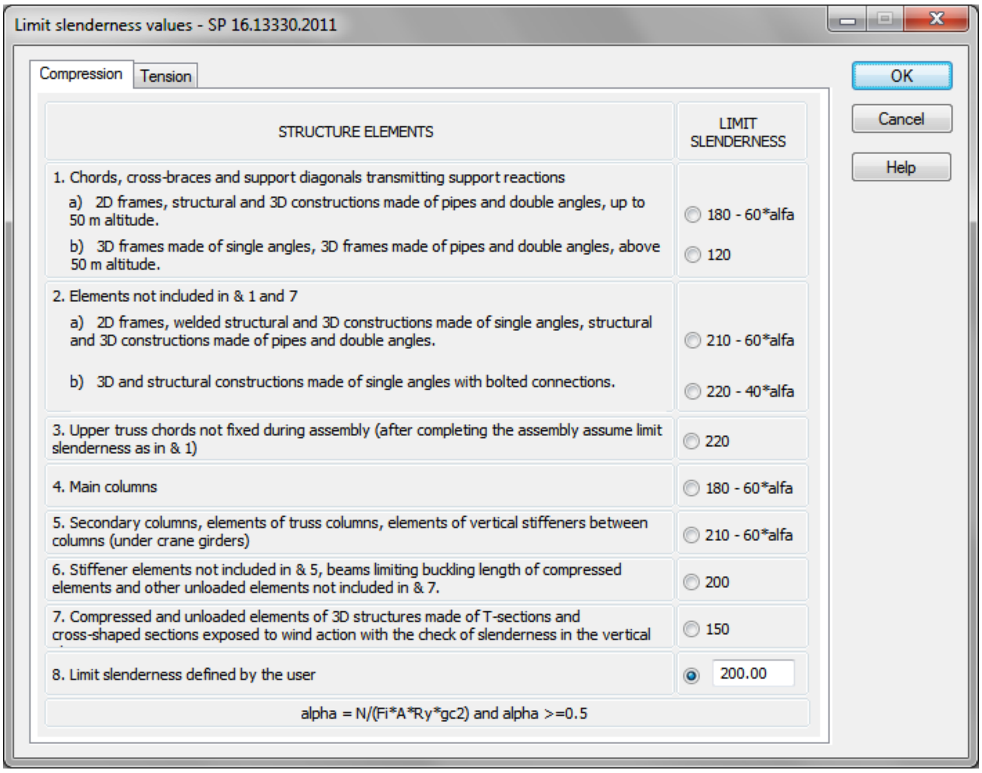


In the CALCULATIONS dialog box set for this task :



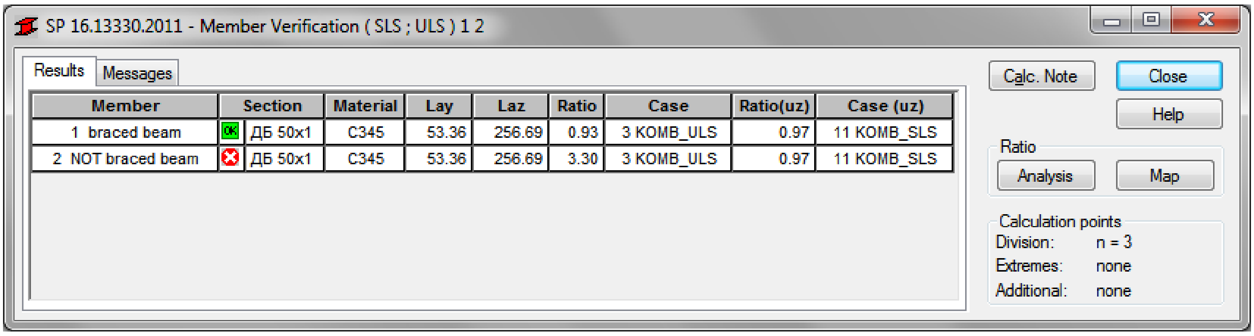


- *Verification option* → Member Verification
- *Loads cases* → n° 3 and n° 11
- *Limit state* → *Ultimate* and *Serviceability* Limit state will be analyzed so switch on both check boxes
- *Material factor* → *gm =1,025 ;* *the "1,025" radio button* is switched on by default in CONFIGURATION dialog box (opened by [Configuration] button)
- *Limit slenderness* → "Defined by the user " on [Compression] tab (radio button selected in LIMIT SLENDERNESS VALUE dialog box which is opened by [Define] button placed in CONFIGURATION dialog box)



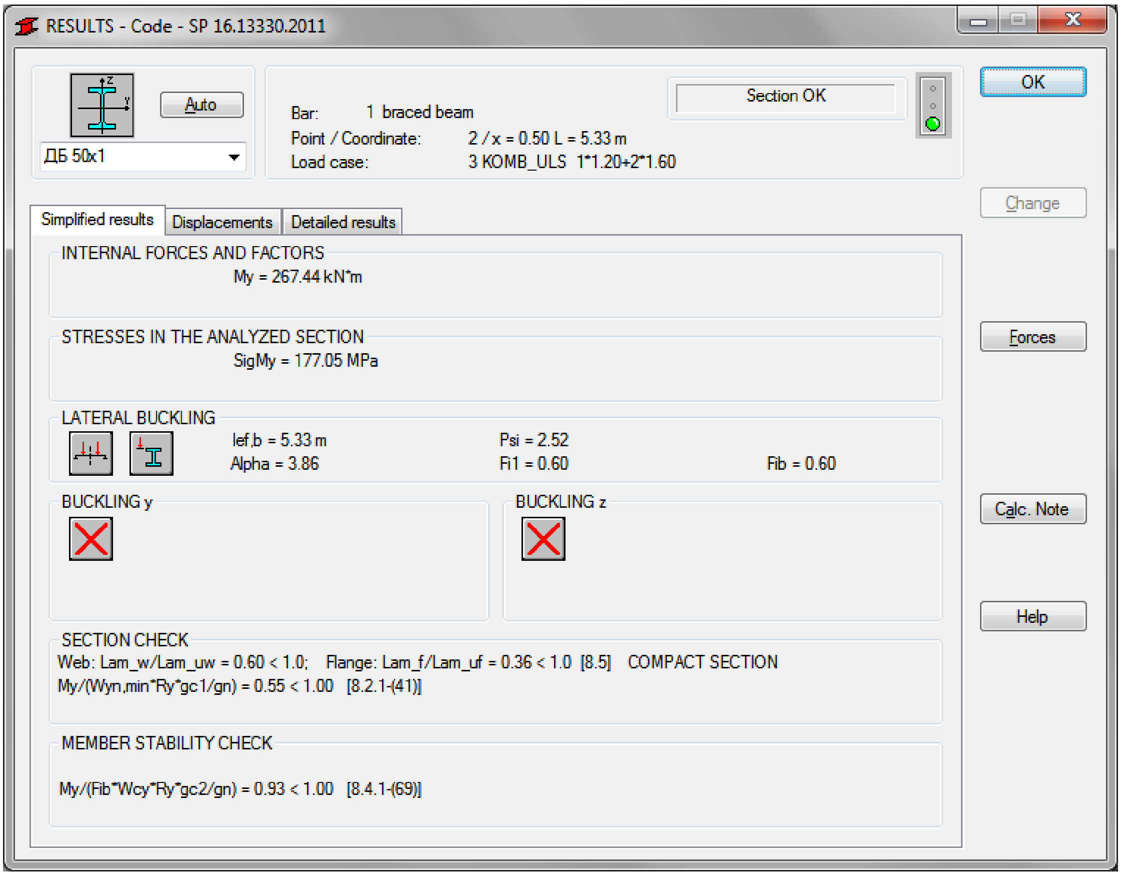
Now, start verifications by pressing *[Calculations]* button.

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

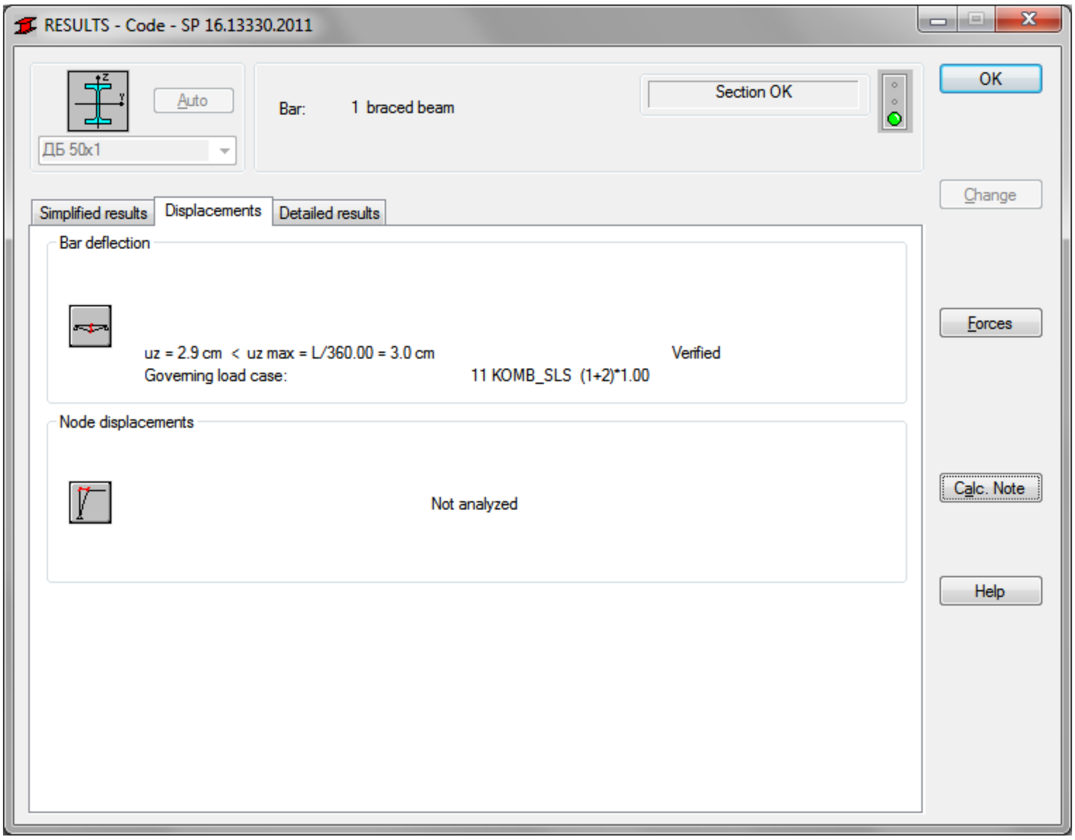


Pressing the line with general results for the member 1 opens the RESULTS dialog box with detailed results for the analyzed member. The view of the RESULTS windows are presented below.

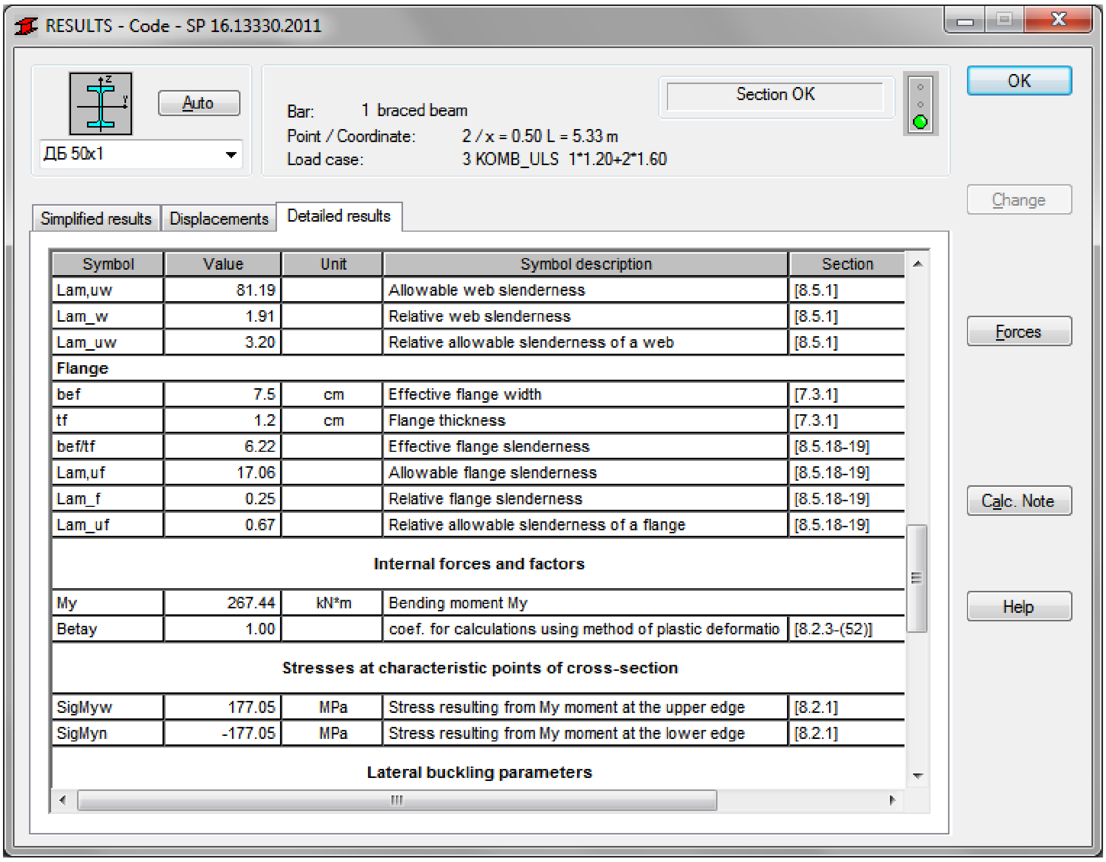
Simplified results tab



Displacements results tab



Detailed results tab



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 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	Rs = 185.60 MPa	Ru = 460.00 MPa	E = 206000.00 MPa
gu = 1.30	gc1 = 1.00	gc2 = 1.00	gn = 1.00

SECTION PARAMETERS: **ДБ 50x1**

ht=49.2 cm			
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:	bef/tf = 6.22	Lam,uf = 17.06	

INTERNAL FORCES AND FACTORS
My = 267.44 kN*m

STRESSES AT CHARACTERISTIC SECTION POINTS
SigMy = 177.05 MPa

LATERAL BUCKLING PARAMETERS:

lef,b = 5.33 m	Psi = 2.52	
Alpha = 3.86	Fi1 = 0.60	Fib = 0.60

BUCKLING PARAMETERS:
About Y axis: About Z axis:

VERIFICATION FORMULAS:
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)]

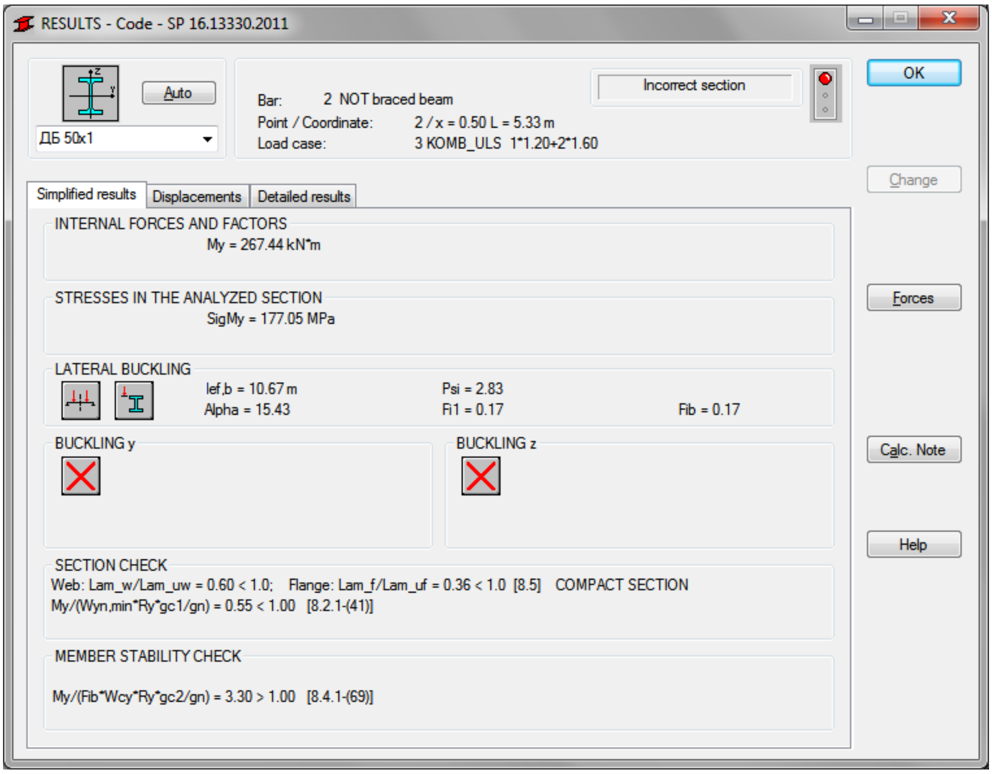
Member stability check
My/(Fib*Wcy*Ry*gc2/gn) = 0.93 < 1.00 [8.4.1-(69)]

LIMIT DISPLACEMENTS
Deflections
uz = 2.9 cm < uz max = L/360.00 = 3.0 cm Verified
Governing Load Case: 11 KOMB_SLS (1+2)*1.00
Displacements *Not analyzed*

Section OK !!!

For comparison the simplified results for the unbraced beam, n° 2 , are presented below.

Simplified results tab



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

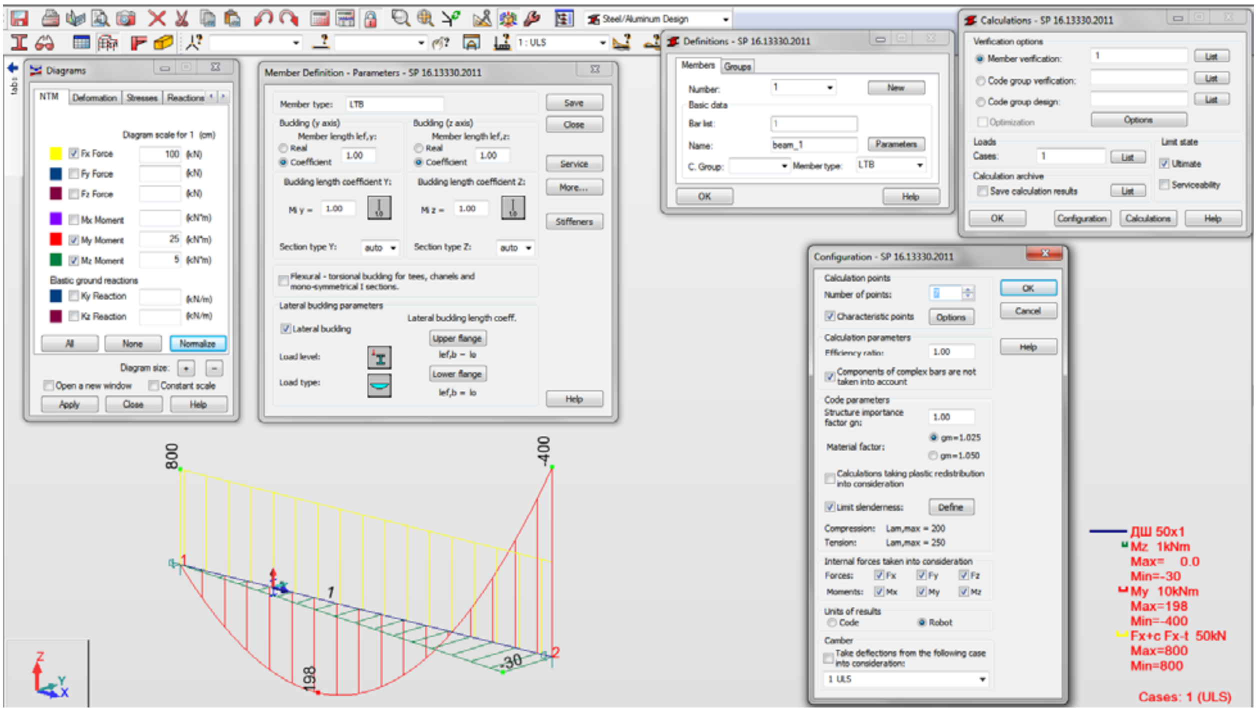
SPECIFICATION:

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
Nc = 800 kN *My* = 400 kNm *Mz* = 30 kNm

Material Properties:

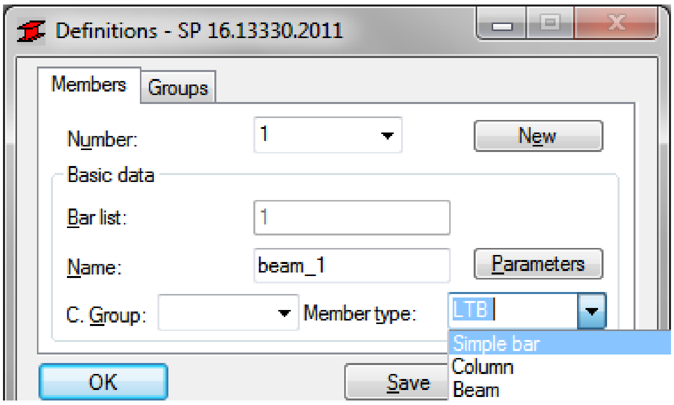
C275 Ry = 270 MPa , R*u* = 370 MPa



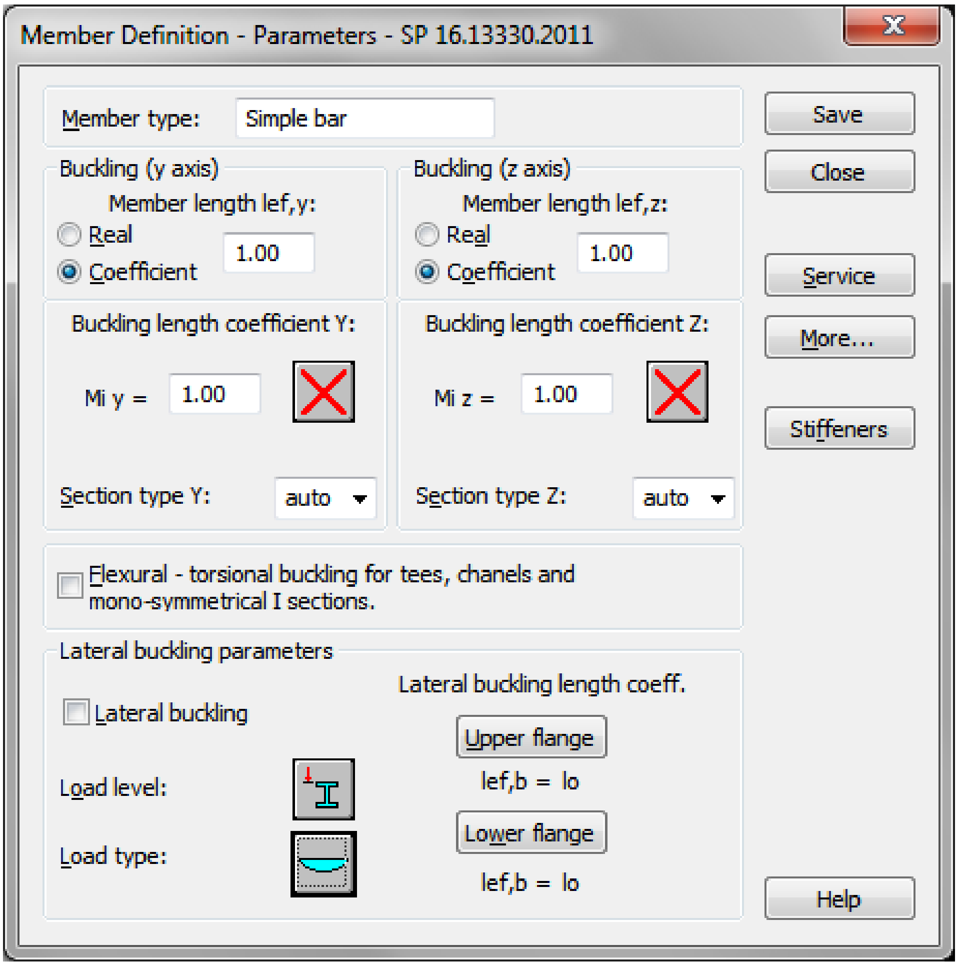
SOLUTION:

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

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

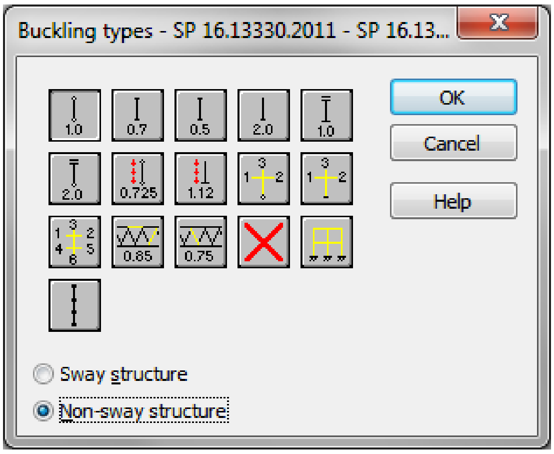


For a chosen member type click the *[Parameters]* button on *Members* tab.
It opens MEMBER DEFINITION–PARAMETERS dialog box.



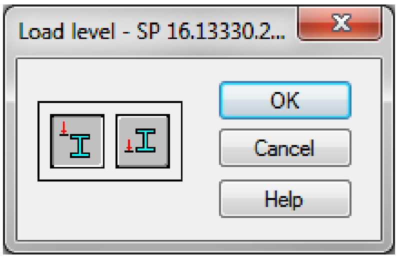
Type a new name in the *Member type* editable field. Then change parameters to meet initial data requirements of the current structure. For this particular task for a bar subjected to a combined loads Nc +My+Mz case :

- 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 ;

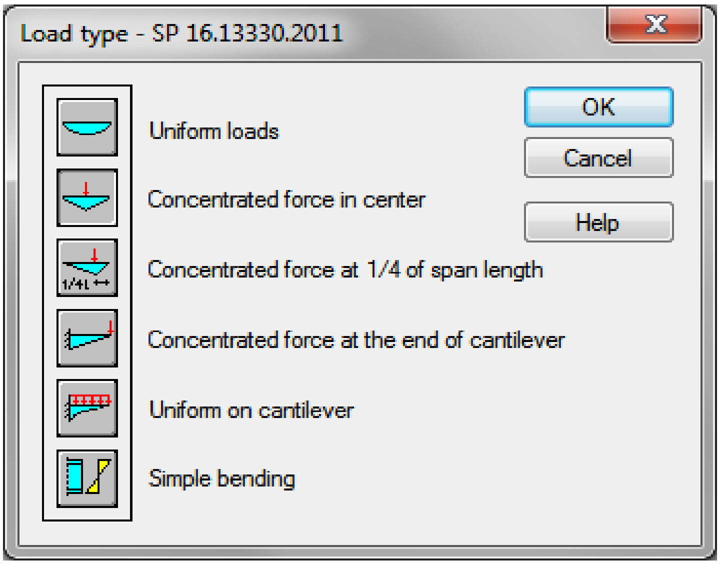


Do the same for the z-z direction.

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

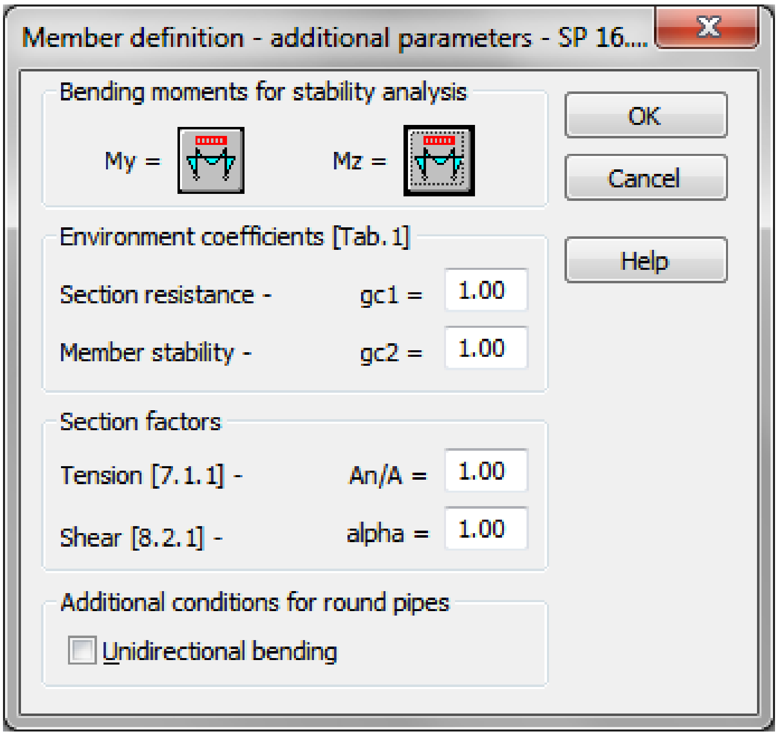


- select appropriate load type by pressing a " load " icon which opens LOAD TYPE dialog box;

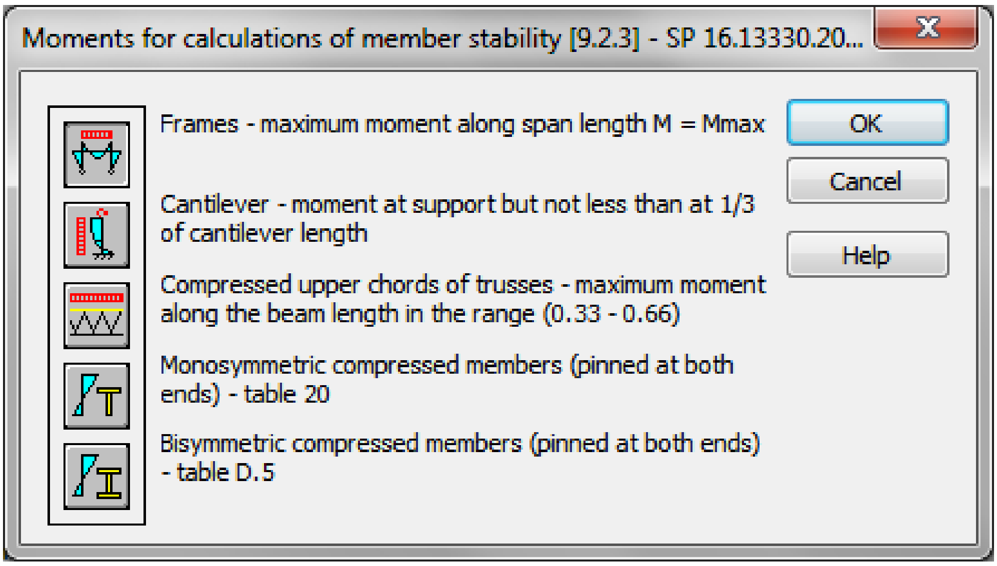


For this task the first icon "uniform loads" was selected.

- 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 ;

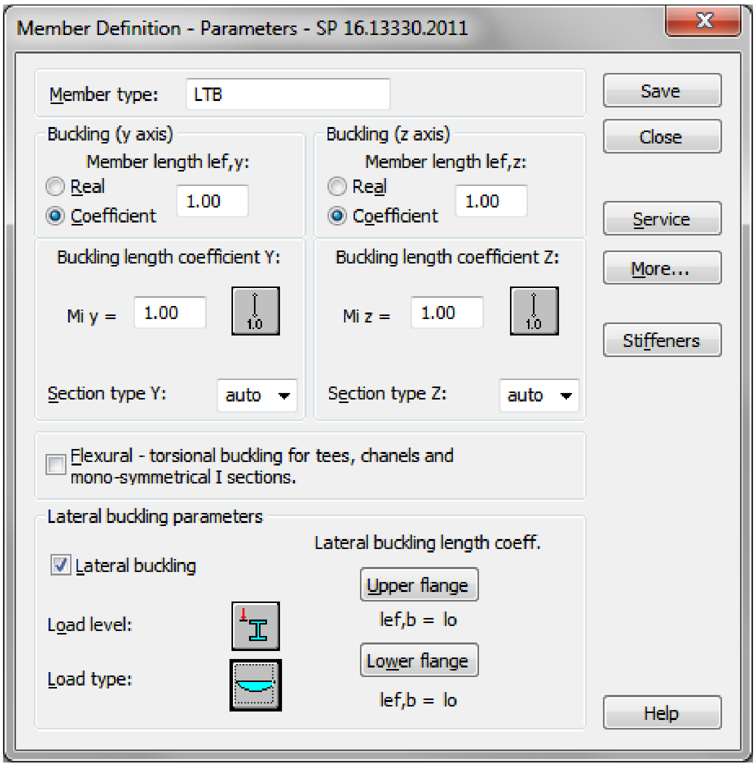


In the newly opened MOMENTS FOR CALCULATIONS OF MEMBER STABILITY dialog box select a needed scheme icon ;



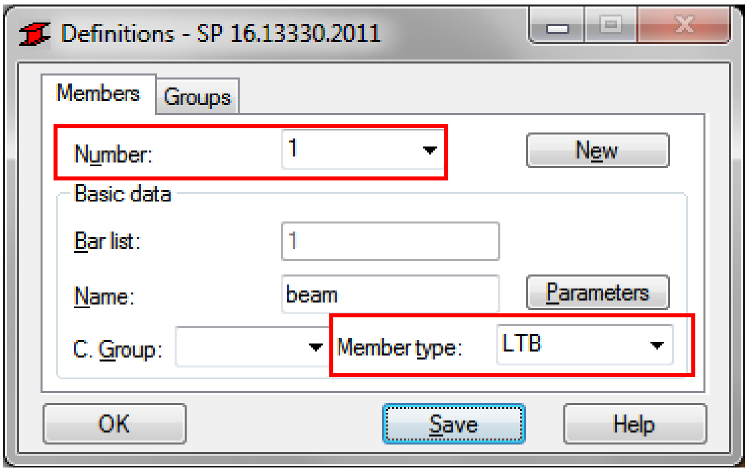
Save the newly-created member type under a new name, e.g. "LTB".

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

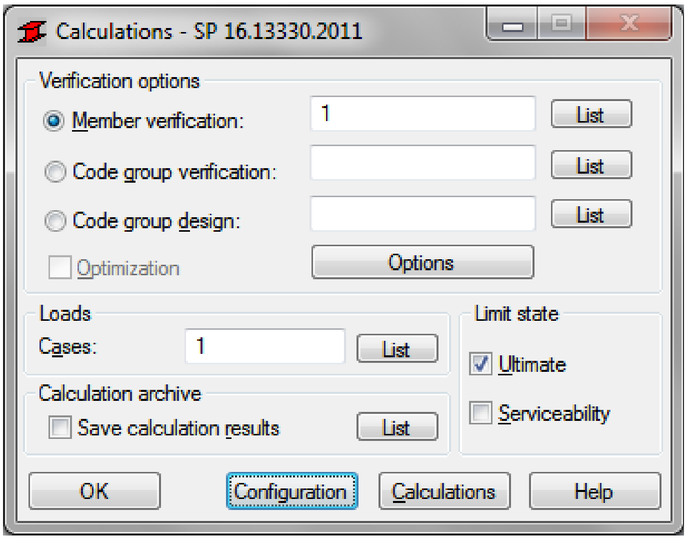


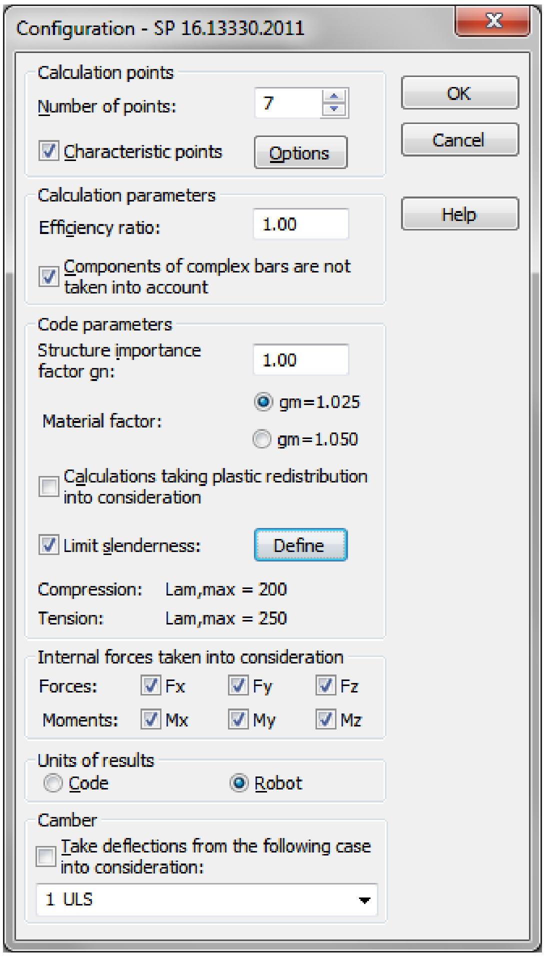
In DEFINITIONS dialog box number of the member must be assigned to the appropriate name of *Member type.*

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



In the CALCULATIONS dialog box set for this task :

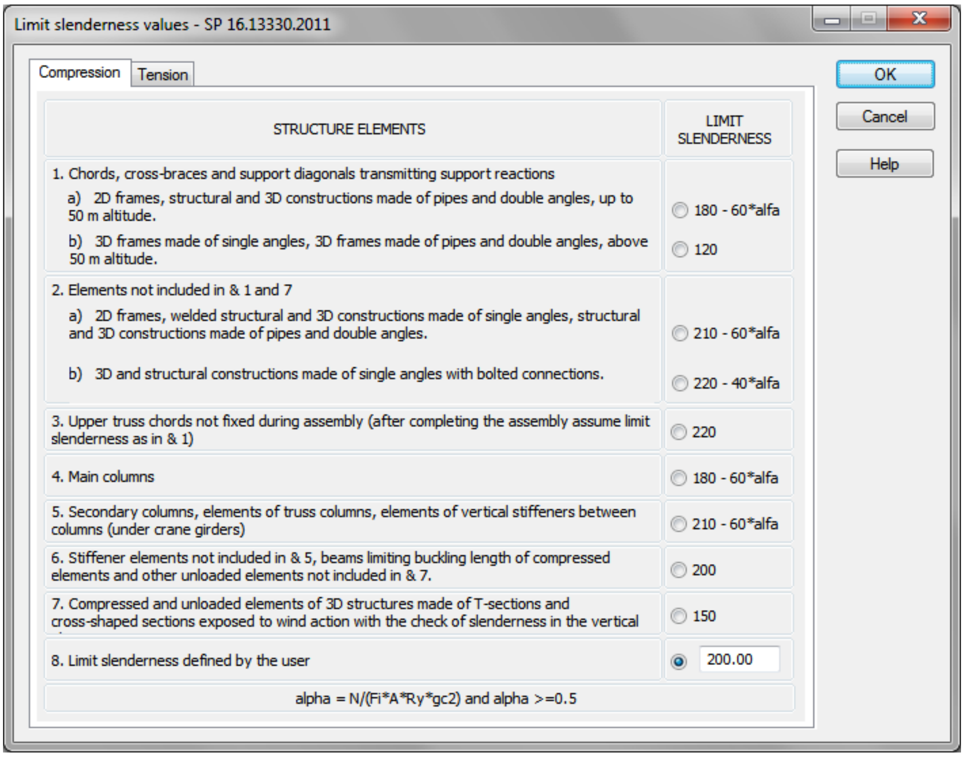




- *Verification option* → Member Verification
- *Loads cases* → n° 1
- *Limit state* → only Ultimate Limit state will be analyzed so switch off *Limit stat Serviceability*.

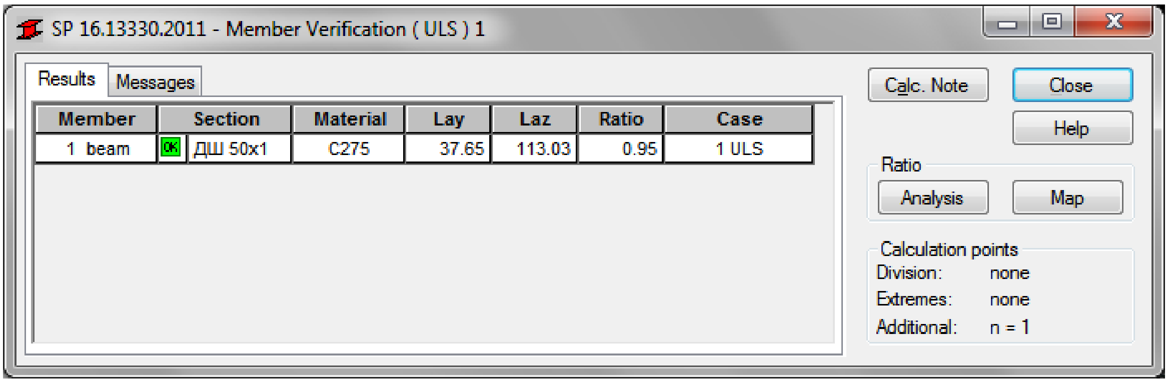
  And in CONFIGURATION dialog box opened by [Configuration] button :

- *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 .



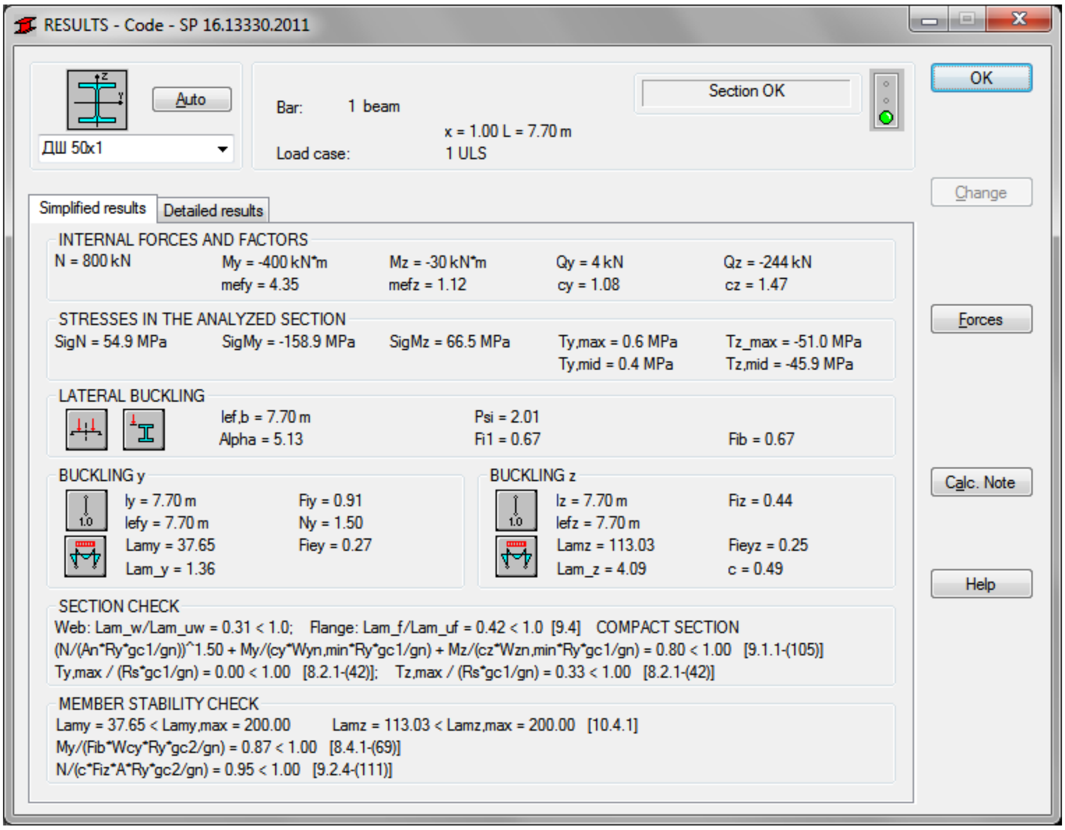
Now, start verifications by pressing *[Calculations]* button.

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

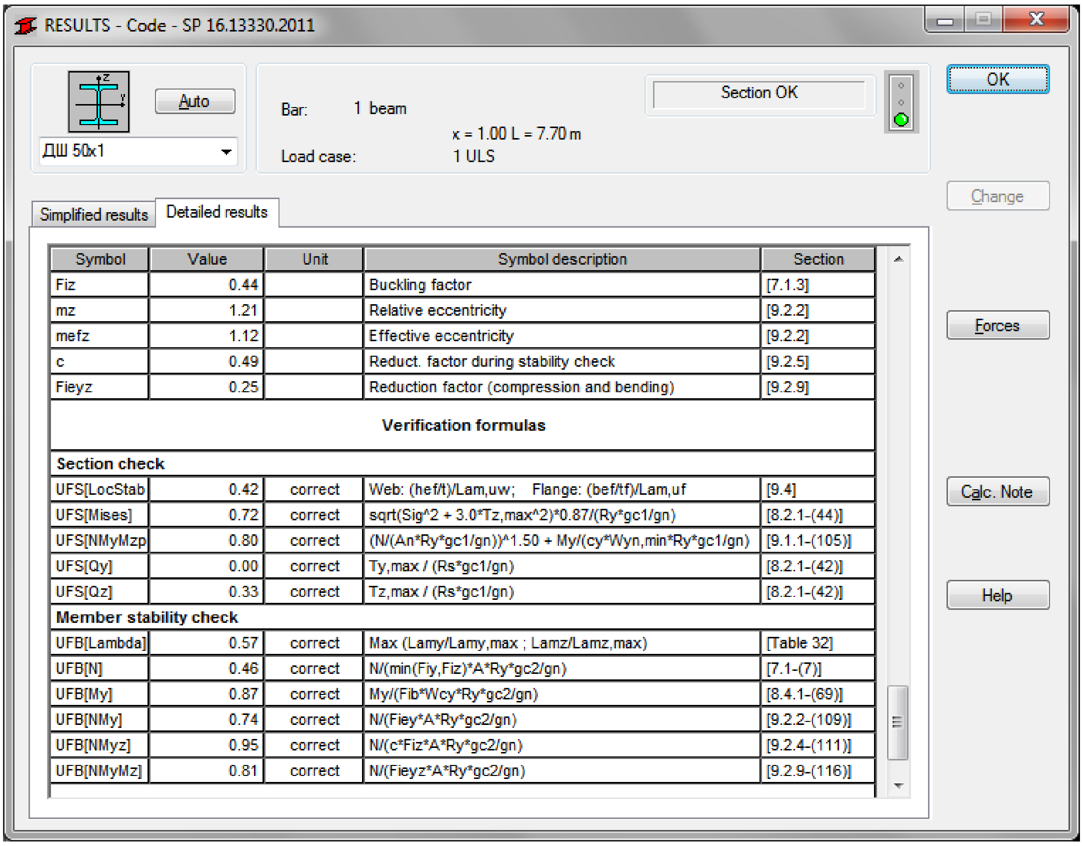


Pressing a line with results for the member 1 opens the RESULTS dialog box with detailed results for the analyzed member. The view of the RESULTS windows are presented below.

Simplified results tab



Detailed results tab



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]
c	0.49		Reduct. factor during stability check	[9.2.5]
Fieyz	0.25		Reduction factor (compression and bending)	[9.2.9]
Verification formulas				
Section check				
UFS[LocStab	0.42	correct	Web: (hef/t)/Lam,uw; Flange: (bef/tf)/Lam,uf	[9.4]
UFS[Mises]	0.72	correct	sqrt(Sig^2 + 3.0*Tz,max^2)*0.87/(Ry*gc1/gn)	[8.2.1-(44)]
UFS[NMyMzp	0.80	correct	(N/(An*Ry*gc1/gn))^1.50 + My/(cy*Wyn,min*Ry*gc1/gn)	[9.1.1-(105)]
UFS[Qy]	0.00	correct	Ty,max / (Rs*gc1/gn)	[8.2.1-(42)]
UFS[Qz]	0.33	correct	Tz,max / (Rs*gc1/gn)	[8.2.1-(42)]
Member stability check				
UFB[Lambda]	0.57	correct	Max (Lamy/Lamy,max ; Lamz/Lamz,max)	[Table 32]
UFB[N]	0.46	correct	N/(min(Fiy,Fiz)*A*Ry*gc2/gn)	[7.1-(7)]
UFB[My]	0.87	correct	My/(Fib*Wcy*Ry*gc2/gn)	[8.4.1-(69)]
UFB[NMy]	0.74	correct	N/(Fiey*A*Ry*gc2/gn)	[9.2.2-(109)]
UFB[NMyz]	0.95	correct	N/(c*Fiz*A*Ry*gc2/gn)	[9.2.4-(111)]
UFB[NMyMz]	0.81	correct	N/(Fieyz*A*Ry*gc2/gn)	[9.2.9-(116)]

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

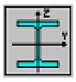
CODE: *SP 16.13330.2011* Steel structures.*
ANALYSIS TYPE: Member Verification

CODE GROUP:
MEMBER: 1 beam **POINT:** **COORDINATE:** 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



SECTION PARAMETERS: ДШ 50x1

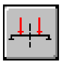
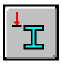
ht=48.4 cm			
bf=30.0 cm	Ay=90.00 cm2	Az=53.24 cm2	Ax=145.70 cm2
t=1.1 cm	Iy=60930.00 cm4	Iz=6762.00 cm4	Ix=88.97 cm4
tf=1.5 cm	Wely=2517.77 cm3	Welz=450.80 cm3	
Web:	hef/t = 36.55	Lam,uw = 117.87	
Flange:	bef/tf = 7.90	Lam,uf = 19.03	

INTERNAL FORCES AND FACTORS

N = 800 kN	My = -400 kN*m	Mz = -30 kN*m
	mefy = 4.35	mefz = 1.12
	Qy = 4 kN	Qz = -244 kN
	cy = 1.08	cz = 1.47

STRESSES AT CHARACTERISTIC SECTION POINTS

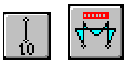
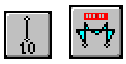
SigN = 54.9 MPa	SigMy = -158.9 MPa	SigMz = 66.5 MPa
	Ty,max = 0.6 MPa	Tz_max = -51.0 MPa
	Ty,mid = 0.4 MPa	Tz,mid = -45.9 MPa

LATERAL BUCKLING PARAMETERS:

lef,b = 7.70 m	Psi = 2.01	
Alpha = 5.13	Fi1 = 0.67	Fib = 0.67

BUCKLING PARAMETERS:

 About Y axis:  About Z axis:

ly = 7.70 m	Fiy = 0.91	lz = 7.70 m	Fiz = 0.44
lefy = 7.70 m	Ny = 1.50	lefz = 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 FORMULAS:
Section check
Web: Lam_w/Lam_uw = 0.31 < 1.0; Flange: Lam_f/Lam_uf = 0.42 < 1.0 [9.4] COMPACT SECTION
(N/(An*Ry*gc1/gn))^1.50 + My/(cy*Wyn,min*Ry*gc1/gn) + Mz/(cz*Wzn,min*Ry*gc1/gn) = 0.80 < 1.00 [9.1.1-(105)]
Ty,max / (Rs*gc1/gn) = 0.00 < 1.00 [8.2.1-(42)]; Tz,max / (Rs*gc1/gn) = 0.33 < 1.00 [8.2.1-(42)]
Member stability check
Lamy = 37.65 < Lamy,max = 200.00 Lamz = 113.03 < Lamz,max = 200.00 [10.4.1]
My/(Fib*Wcy*Ry*gc2/gn) = 0.87 < 1.00 [8.4.1-(69)]
N/(c*Fiz*A*Ry*gc2/gn) = 0.95 < 1.00 [9.2.4-(111)]

Section OK !!!