S.C.R. PIEMONTE S.p.A.
Società di Committenza Regione Piemonte
corso Marconi 10 - 10125 Torino
tel +39 011 6548311 - fax +39 011 6694665

OPERE PUBBLICHE

NOME DELLA PROVINCIA	NOME DEI COMUNI/ASL
PROVINCIA DI TORINO	**COMUNE DI TORINO**

SERVIZIO/LIVELLO PROGETTUALE

PROGETTO ESECUTIVO

CODICE OPERA	TITOLO INTERVENTO
15L65U01A	**Lavori di manutenzione straordinaria ed efficientamento energetico del palasport olimpico Pala Alpitour - FASE1**

	TITOLO ELABORATO
Tavola n. **006**	Validazione codice di calcolo

DATA	SCALA	AREA PROGETTUALE
27/10/2017	**-**	**GE**

CODICE GENERALE ELABORATO

15L65U01A_0_0_E_GE_01_MG_006_0

NOME FILE

TAV_N006_15L65U01A_0_0_E_GE_01_MG_006_0_Validazione_Codici_Calcolo.pdf

VERSIONE	DATA	DESCRIZIONE
0	27 ottobre 2017	Prima redazione

PROGETTISTA	TIMBRI - FIRME
MDR ENGINEERING S.R.L. TORINO - via Caboto, 53 - 10129 - tel. 011-19703428 - fax 011-19703432 e-mail: studio@durbanom.it PROGETTO STRUTTURALE: Ing. Marcello Durbano - ord. Ing. Cuneo n° 1079	Responsabile del progetto: ing. Marcello Durbano ORDINE DEGLI INGEGNERI DELLA PROVINCIA DI CUNEO 1079 Dott. Ing. Marcello Durbano

AFFIDATARIO	TIMBRI - FIRME
(ragione sociale, e eventuale logo)	Direttore Tecnico: ....

ORGANISMO DI CONTROLLO	S.C.R. PIEMONTE S.P.A.
Responsabile di Commessa:	Responsabile del Procedimento: arch. Sergio MANTO

Sommario

Presentazione del codice ci calcolo

Il programma di calcolo utilizzato è il Robot Structural Analysis Professional 2017 fornito dalla Autodesk

GRADO DI AFFIDABILITA' DEL CODICE

L' affidabilita' del codice di calcolo e' garantita dall'esistenza di un ampia documentazioe di supporto, come indicato nel paragrafo precedente. La presenza di un modulo CAD per l'introduzione di dati permette la visualizzazione dettagliata degli elementi introdotti. E' possibile inoltre ottenere rappresentazioni grafiche di deformate e sollecitazioni della struttura. Al termine dell'elaborazione viene inoltre valutata la qualita' della soluzione, in base all'uguaglianza del lavoro esterno e dell'energia di deformazione.

MOTIVAZIONE DELLA SCELTA DEL CODICE

Il codice permette in campo elastico lineare un'analisi dettagliata del comportamento dell'intera struttura, tenendo conto del comportamento irrigidente di setti anche complessi e solai considerati con la loro effettiva rigidezza. E' possibile inoltre scegliere il grado di affinamento dell'analisi di elementi complessi utilizzando mesh via via piu' dettagliate.

ESAME DEI RISULTATI E CONTROLLI

VALUTAZIONE DELLA CORRETTEZZA DEL MODELLO

Il modello di calcolo adottato e' da ritenersi appropriato in quanto non sono state riscontrate labilita', le reazioni vincolari equilibrano i carichi applicati, la simmetria di carichi e struttura da' origine a sollecitazioni simmetriche.

GIUDIZIO MOTIVATO DI ACCETTABILITA' DEI RISULTATI

L'analisi critica dei risultati e dei parametri di controllo nonche' il confronto con calcolazioni di massima eseguite manualmente porta ad confermare la validita' dei risultati.

Il sistema Autodesk *Robot Structural Analysis Professional 2017* è un programma grafico integrato che serve a modellare, analizzare e dimensionare le strutture di vari tipi. Lavora in ambiente Windows. Consente la modellazione di elementi finiti tipo barra o elementi finiti superficiali per la generazione della mesh delle strutture tipo piastra-guscio.

CONVENZIONI SUI SEGNI

Per le convenzioni sui segni, la direzione positiva delle forze e degli spostamenti è ugauale alle direzioni positive degli assi. Le direzioni positive degli angoli, delle rotazioni o dei momenti nel sistema di coordinate sia locale che globale vengono stabilite in base alla regola della vite destrorsa. Tale convenzione definisce i segni delle azioni esterne, azioni nodali, quelle degli spostamenti e delle rotazioni. La convenzione dei segni negli elementi barra è basata sulla convenzione delle forze di sezione (le forze di sezione hanno lo stesso segno se, all'estremità della barra, hanno lo stesso effetto delle forze nodali positive applicate al nodo iniziale della barra)

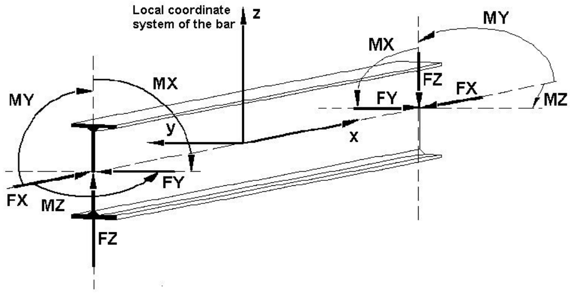
CARICHI E VINCOLI
I carichi possono essere applicati sia ai nodi sotto forma di forze o coppie concentrate, sia sulle aste come forze distribuite, trapezie, concentrate, come coppie e come distorsioni termiche. I carichi possono essere di tipo statico o mobile.
Per i vincoli esterni consente la definizione di vincoli di tipo rigido (con possibilità di definire il sollevamento del vincolo stesso), elastico e con smorzamento. Per i vincoli interni è possibile definire i gradi di libertà di ogni singolo nodo.

ANALISI DELLA STRUTTURA E RISULTATI
Il sistema *Autodesk Robot Structural Analysis Professional 2017* consente all'utente di definire i parametri di dieversi tipi di analisi della struttura: statica lineare, statica non lineare, analisi modale, analisi armonica, analisi sismica ecc.
Al termine del calcolo i risultati dell'analisi possono essere visualizzati in forma grafica o tabellare con varie possibilità di approfondimento di dettaglio.
A supporto del programma e' fornito un ampio manuale d'uso .

Licenza RSAPRO_F_S 547D1

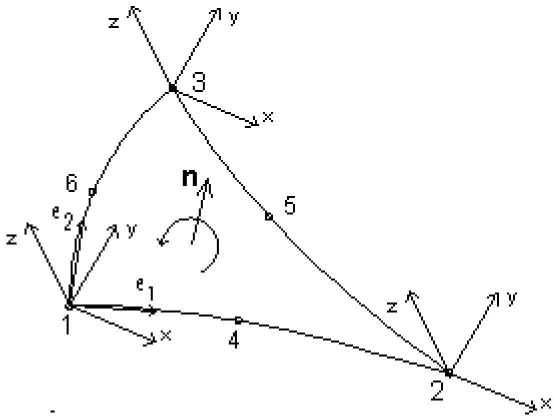
Convenzione dei segni

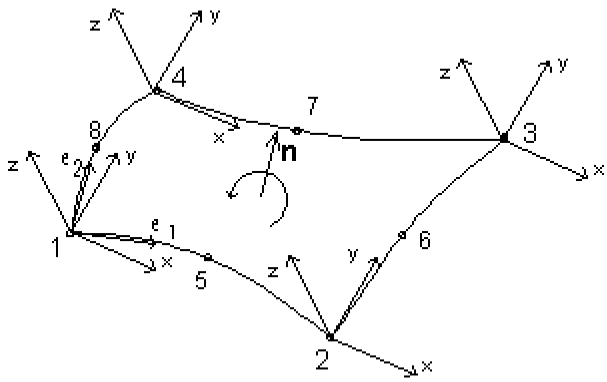
L'orientamento positivo delle forze e degli spostamenti sono concordi con la direzione positiva degli assi del sistema di coordinate. L'orientamento positivo degli angoli, delle rotazioni, o momenti nel sistema di coordiante locale o globale sono determinate in base alla regola della mano destra. Questa convenzione definisce il segno delle forze esterne, delle forze nodali, degli spostamenti e delle rotazioni. Questi valori sono utilizzati nella definizione della struttura, durante il calcolo e nell'esplicitazione dei risultati. La convenzione dei segni per le barre è basata sulla convenzione usata per le forze di sezione. Secondo questa regola le forze di sezione hanno lo stesso segno se all'estremità della barra fanno lo stesso effetto delle forze nodali positive applicate al nodo iniziale della barra (forze la cui direzione coincide con quella dell'asse del sistema di coordinate locale). Per questo le forze di compressione sono positive e le forze di trazione sono negative. I momenti flettenti positivi MY causano una trazione di quelle fibre della trave che sono presenti nella parte negativa dell'asse di coordinate locale "z". I momenti flettenti positivi MZ causano una trazione di quelle fibre della trave che sono presenti dalla parte positiva dell'asse di coordinate "y".
Le direzioni positive delle forze per questa convenzione dei segni sono presentate nella figura sotto:



Ogni nodo ha un proprio sistema locale; quindi, il sistema di coordinate locali per un intero elemento finito non è richiesto. È importante conoscere l'orientamento del vettore normale sulla superficie dell'elemento finito. Preferibilmente, i vettori normali per tutti gli elementi finiti dovrebbero avere lo stesso orientamento. In caso contrario, la definizione del carico per un elemento (come la pressione perpendicolare alla superficie del guscio) potrebbe essere errata.

L'orientamento di un vettore normale (perpendicolare alla superficie di un elemento guscio) viene determinato in base alla regola di destra (nel senso del nodo primo-ultimo dell'elemento). Nella figura seguente sono mostrati i sistemi di coordinate locali e l'orientamento del vettore normale rispetto alla superficie dell'elemento, presentando elementi finiti a 6 nodi e 8 nodi.

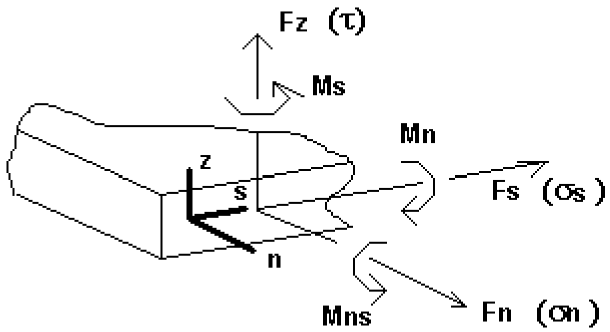




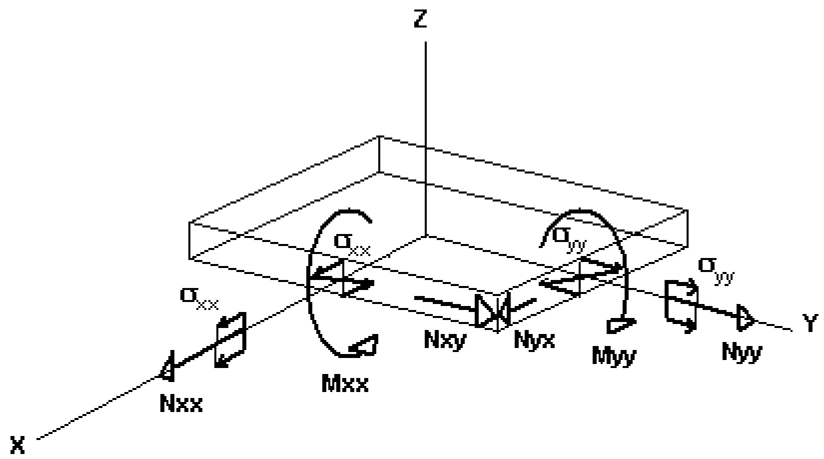
Per elementi finiti piani si ottengono forze e sollecitazioni in un elemento. Sono determinati in base alla posizione rispetto al vettore normale locale e alla tangente alla sezione trasversale. Sono adottate le seguenti convenzioni di segno:

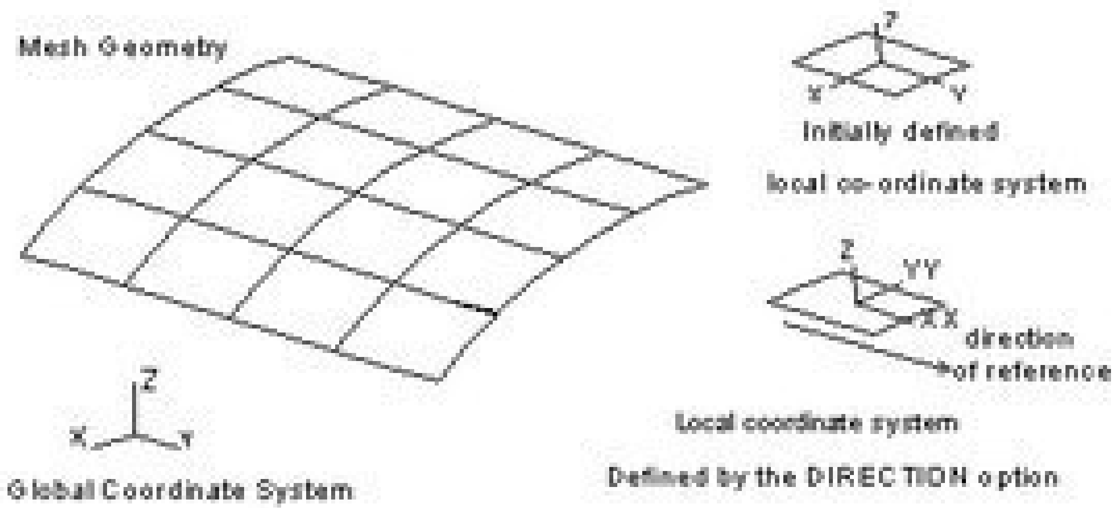
n – vettore perpendicolare alla sezione dell’elemento
s – vettore parallelo alla sezione dell’elemento
z – vettore normale appartenente alla sezione dell’elemento e perpendicolare al vettore s

Questi vettori costituiscono un sistema di coordinate cartesiane a destra. L'orientamento positivo delle forze, dei momenti e delle sollecitazioni in una determinata sezione trasversale è definito in base all'orientamento dei vettori n / s / z. Questo è rappresentato schematicamente nella figura seguente. Le forze, i momenti e le sollecitazioni mostrate nella figura presentano segni positivi.



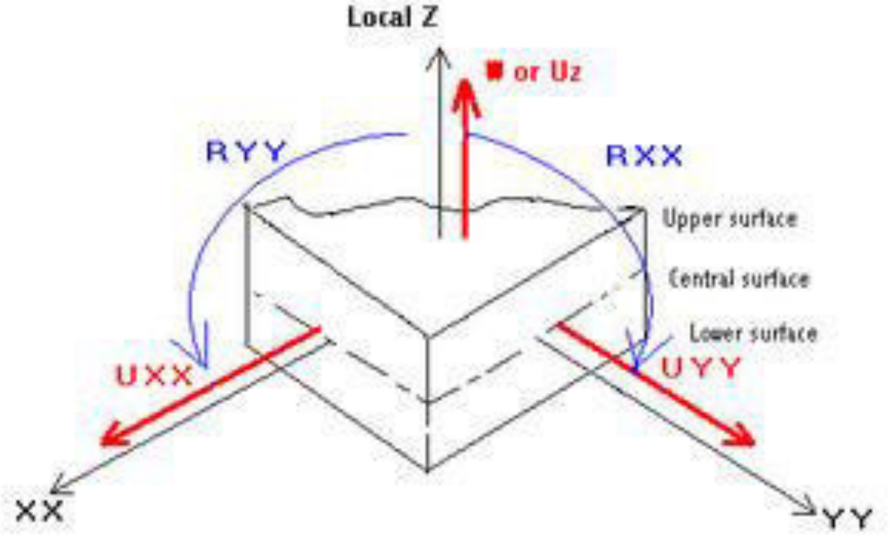
I risultati ottenuti per FE piani vengono presentati in sistemi di coordinate locali che possono essere definiti e modificati dall'utente in qualsiasi momento della presentazione dei risultati. Ad esempio, l'orientamento positivo delle forze e le corrispondenti sollecitazioni in un nodo vengono presentate per l'asse X come direzione di riferimento.



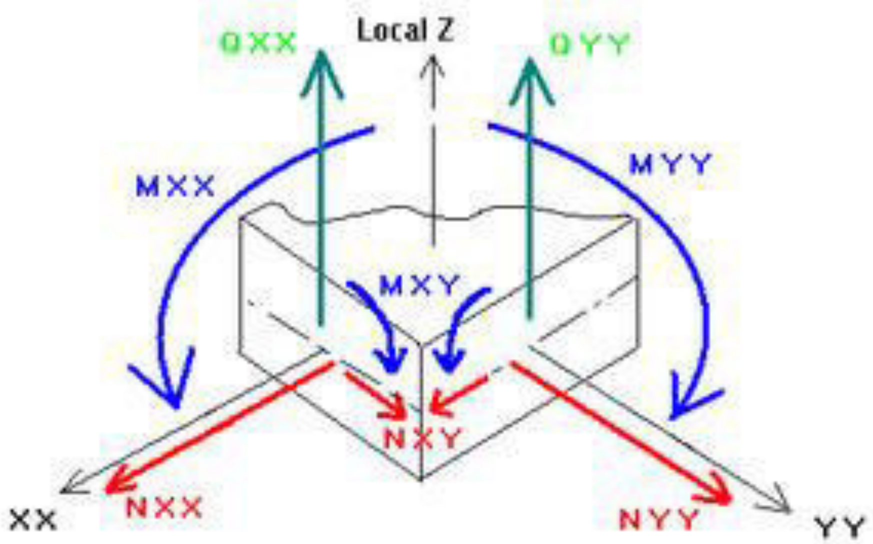


The following define the sign convention applied to forces for planar finite elements.

Sign convention for displacements and rotation



Sign convention for forces



Autodesk®
Robot™ Structural Analysis Professional

VERIFICATION MANUAL

Version 5.2, March 2014

according to:

“***Guide de validation des progiciels de calcul de structures***”
AFNOR, 1990

INTRODUCTION

This verification manual contains a range of static and dynamic benchmark tests covering fundamental types of behaviour encountered in structural analysis. 58 examples of static, dynamic, and thermo-mechanics problems are solved using bar, plate, and shell FE. All the examples have been taken from:

"Guide de validation des progiciels de calcul de structures" **AFNOR, 1990.**

Benchmark results (signed as “AFNOR”) were recalled, and originally compared with results of **Autodesk Robot Structural Analysis Professional 2013** (signed further as “Robot”). The comparison of results is still valid for the next Robot versions.

Each problem contains the following parts:

- the name of the benchmark as used in the AFNOR guide,
- short problem description,
- scheme of the model,
- comparison between Robot results and reference values.

STATIC ANALYSIS

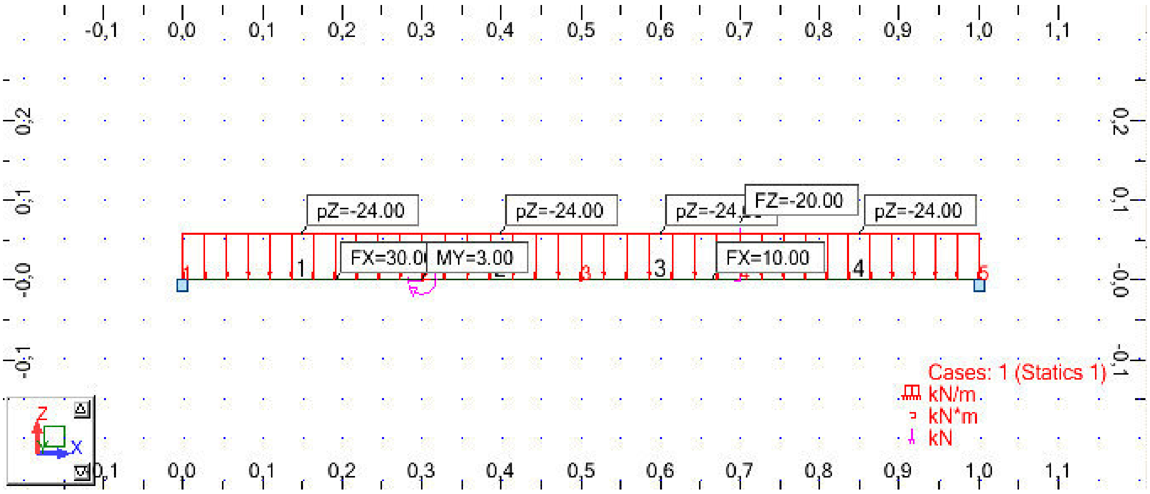
1. BAR STRUCTURES

VERIFICATION EXAMPLE

2D Euler's beam bending - SSLL01/89

Name of the test: SSLL01/89
Reference: AFNOR
Specification: Uni-directional bending of an elastic (Euler) beam

GEOMETRY: Length: L = 1,0 m, fixed ends
Section: Iz=1.7e-8, E=2.1e11



DATA FILE: SSLL01.str (in English)

COMPARISON:

Node	Compared result	Value		Difference %
		Robot *(Robot results)*	**AFNOR** *(Referenced values)*	
3	Shearing force (N)	540	-540	0.0
3	Bending moment (Nm)	2800	2800	0.0
3	Vertical displacement (m)	-4.90196e-2	-4.90196e-2	0.0
1	Horizontal reaction (N)	-24000	-24000	0.0

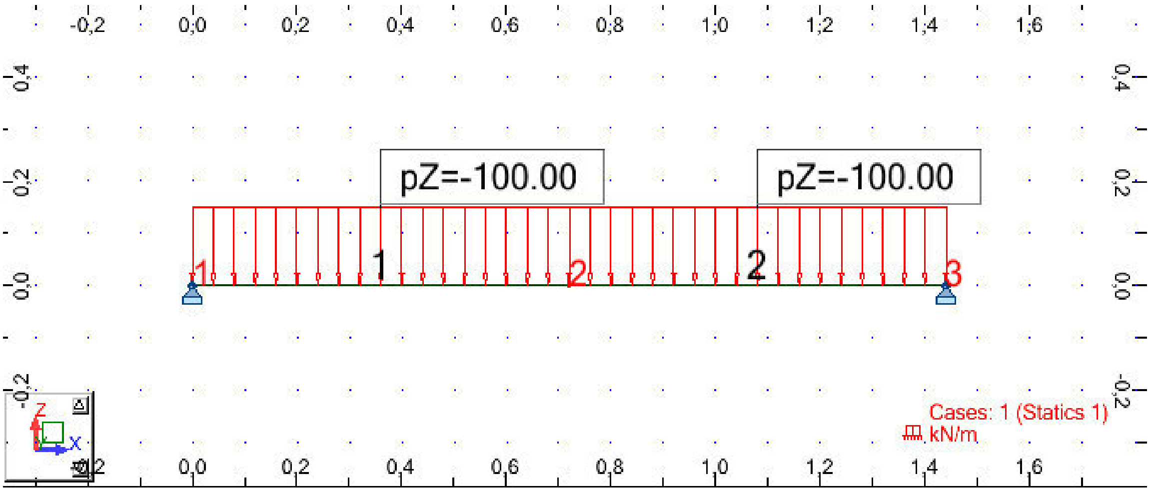
CONCLUSIONS:

Exact agreement of results.
The different signs of shear forces arise from different local coordinates sign convention.

VERIFICATION EXAMPLE

2D Timoshenko's beam bending - SSLL02/89

Name of the test: SSLL02/89
Reference: AFNOR
Specification: Influence of shearing stresses (Timoshenko's beam)
Elastic, linear, isotropic material
GEOMETRY: Length: L = 1,44 m, simply supported ends
Section: Iz=2810e-8, Ax=31e-4, Ay=(31E-4/2.42), E=2.1e11



DATA FILE: SSLL02.str (in English)

COMPARISON:

Node	Compared result	Value		Difference %
		Robot	AFNOR	
2	displacement (m)	-1.25926e-3	-1.25926e-3	0.0

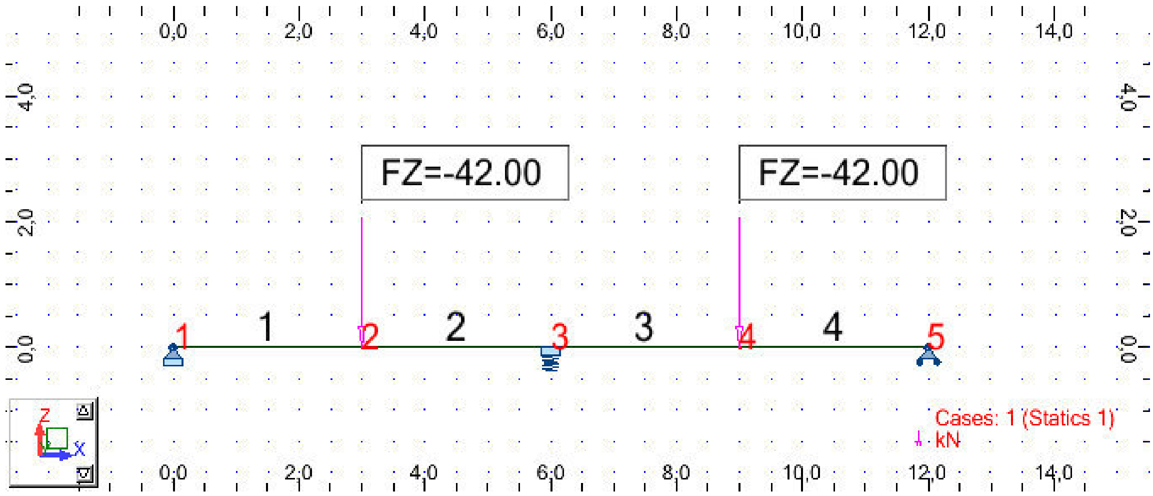
CONCLUSION:

Exact agreement of results.

VERIFICATION EXAMPLE

Beam with elastic support - SSLL03/89

Name of the test: SSLL03/89
Reference: AFNOR
Specification: Simple beam under bending with elastic support in the center of length; material: elastic, linear, isotropic.
GEOMETRY: Length: L = 12 m, simply supported at ends and in the middle
Section: Iz=6.3 e-4, E=2.1e11
Stiffness Kz=2.1 e6 N/m.



DATA FILE: SSLL03.str (in English)

COMPARISON:

Node	Compared result	Value		Difference %
		Robot	AFNOR	
3	Bending moment (Nm)	63000	63000	0.0
3	Displacement UZ (m)	-0.010	-0.010	0.0
3	Vertical reaction (N)	21000	21000	0.0

CONCLUSION:

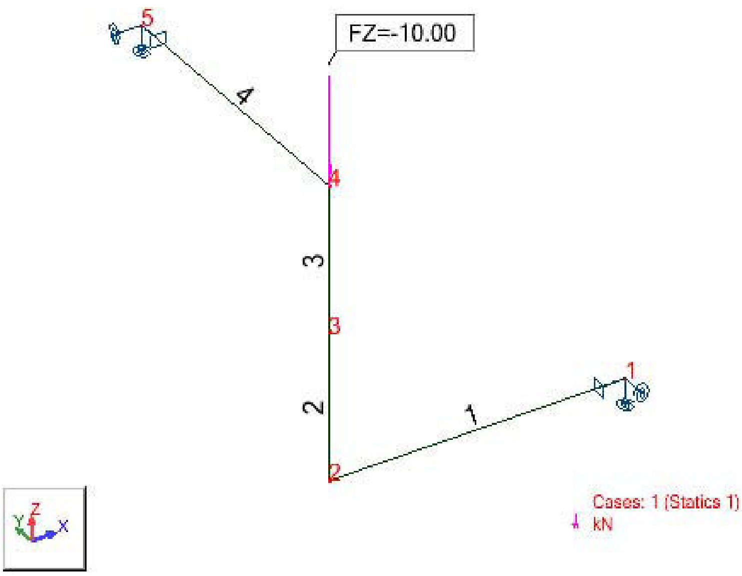
Exact agreement of results.

VERIFICATION EXAMPLE

3D frame with elastic supports - SSLL04/89

Name of the test: SSLL04/89
Reference: AFNOR
Specification: Spatial bar with elastic supports, under bending and torsion; material: elastic, linear, isotropic (non-compressible bars assumed)

GEOMETRY:



DATA FILE: SSLL04.str

COMPARISON:

Node	Compared result	Robot	AFNOR	Difference %
5	Moment MX (Nm)	1562.5	1562.5	0.0
5	Moment MY (Nm)	-8437.5	-8437.5	0.0
5	Moment MZ (Nm)	3125.0	3125.0	0.0
1	Moment MX (Nm)	-1562.5	-1562.5	0.0
1	Moment MY (Nm)	-8437.5	-8437.5	0.0
1	Moment MZ (Nm)	3125.0	3125.0	0.0
5	Displacement UY (m)	-0.029762	-0.029762	0.0
5	Rotation RX (rad)	0.16071	0.16071	0.0
5	Displacement UZ (m)	-0.37004	-0.37004	0.0

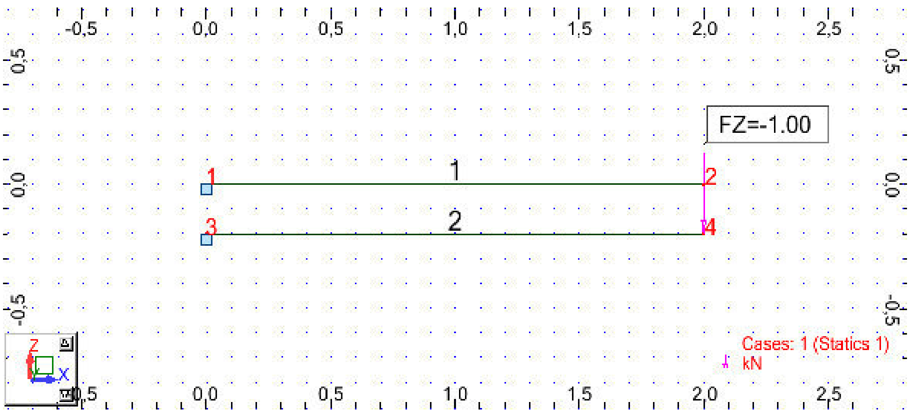
CONCLUSIONS:

Exact agreement of results.

VERIFICATION EXAMPLE

Bending of rigidly connected beams - SSLL05/89

Name of the test: SSLL05/89
Reference: AFNOR
Specification: Beams with rigid link – bending of non-compressible bars
GEOMETRY: Length: L = 2 m, distance 0,2 m,
Left ends - fixed, right – rigidly linked
Section: Iz=4/3e-8, Ax=1.0, E=2e11



DATA FILE: **SSLL05.str**

COMPARISON:

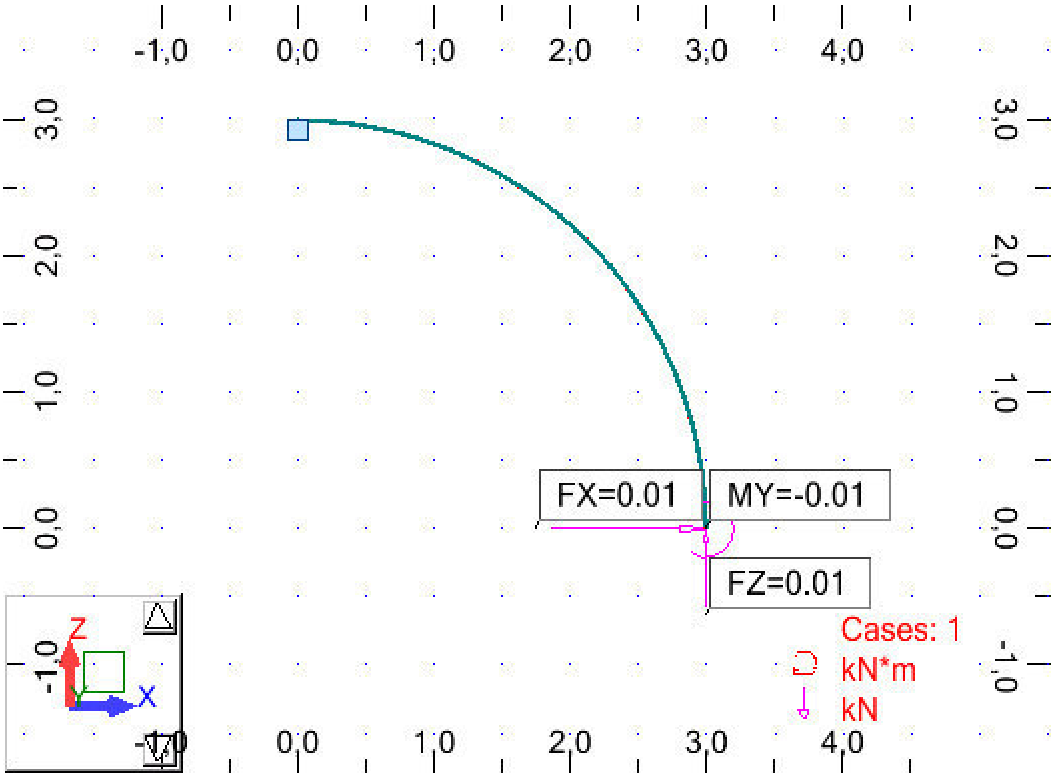
Node	Compared result	Robot	AFNOR	Difference %
2	Displacement (m)	-0.125	-0.125	0.0
4	Displacement (m)	-0.125	-0.125	0.0
1	Vertical reaction (N)	500	500	0.0
1	Moment My (Nm)	-500	500	0.0
3	Vertical reaction (N)	-500	500	0.0
3	Moment My (Nm)	-500	500	0.0

CONCLUSION:
Exact agreement of results
(taking into account different sign convention).

VERIFICATION EXAMPLE

2D circular arch bending - SSLL06/89

Name of the test: SSLL06/89
Reference: AFNOR
Specification: Bending of a quarter circle, fixed at the end, made from hollow round section; elastic linear material, non-compressible bars assumed
GEOMETRY: Radius of the circle = 3 m, upper end fixed.
Section: R=10mm, thickness=2mm (Ax=1.0), E=2e11



DATA FILE: SSLL06.str

COMPARISON:

Node	Compared result	Robot	AFNOR	Difference %
91	Displacement UX(m)	0.3791	0.3791	0.0
91	Displacement UZ(m)	0.2417	0.2417	0.0
91	Rotation RY (rad)	0.1654	0.1654	0.0

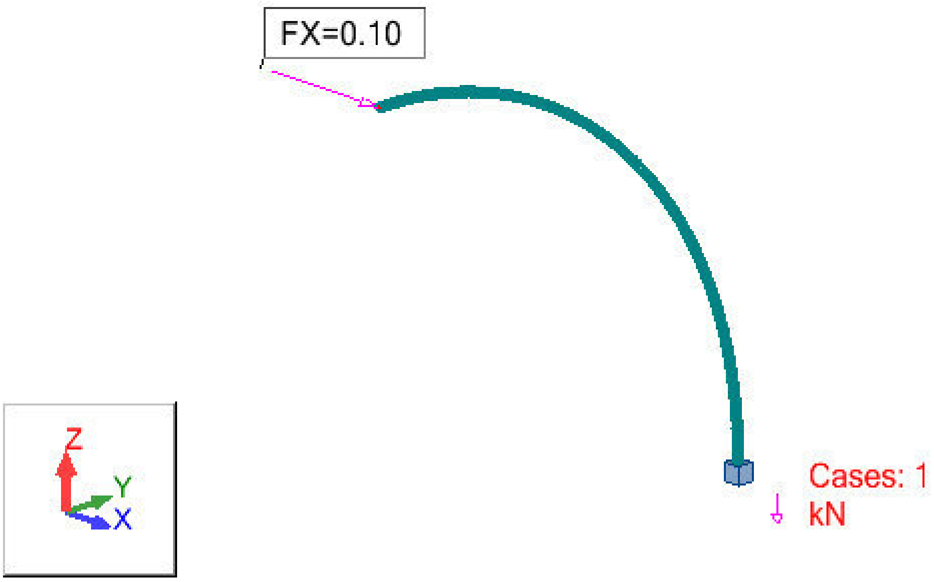
CONCLUSION:

Exact agreement of results.

VERIFICATION EXAMPLE

3D circular arch transverse bending - SSLL07/89

Name of the test: SSLL07/89
Reference: AFNOR
Specification: A quarter circle fixed at the end, bending and torsion of a thin-walled (hollow) round section; material elastic linear isotropic.
GEOMETRY: Radius of the circle = 1 m, lower end fixed.
Section: R=10mm, thickness=2mm, E=2e11



DATA FILE: SSLL07.str

COMPARISON:

Node	Compared result	Robot	AFNOR	Difference %
91	Displacement UX (m)	0.13461	0.13462	0.0
16	Moment MX (Nm)	74.115 (mean)	74.118	0.004
16	Moment MZ (Nm)	96.589 (mean)	96.592	0.003

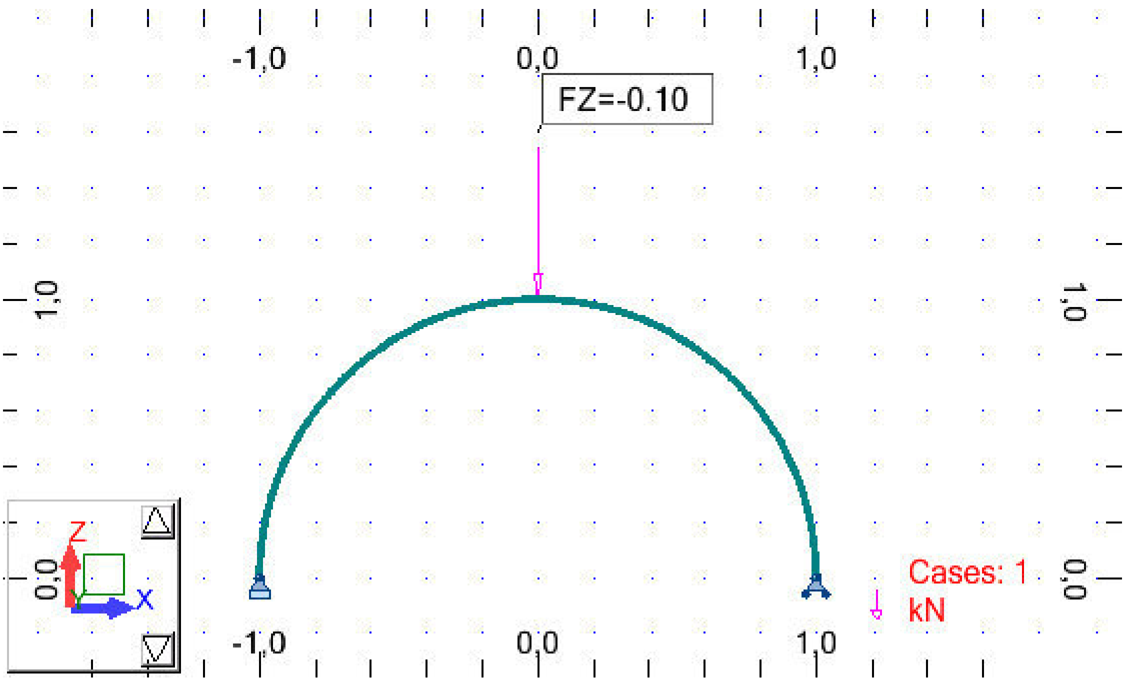
CONCLUSIONS:

Excellent agreement of results.

VERIFICATION EXAMPLE

2D semi-circular arch bending - SSLL08/89

Name of the test: SSLL08/89
Reference: AFNOR
Specification: A half circle simply supported at the ends, bending of a thin-walled (hollow) round section; material elastic linear isotropic.
GEOMETRY: Radius of the circle = 3 m, upper end fixed.
Section: R=10mm, thickness=2mm (Ax=1.0), E=2e11



DATA FILE: SSLL08.str

COMPARISON:

Node	Compared result	Robot	AFNOR	Difference %
1	Rotation RY(rad)	3.0774e-2	-3.0774e-2	0.0
181	Rotation RY(rad)	-3.0774e-2	3.0774e-2	0.0
91	Displacement UZ (m)	-1.9206e-2	-1.9206e-2	0.0
181	Displacement UX (m)	5.3912e-2	5.3912e-2	0.0

CONCLUSIONS:

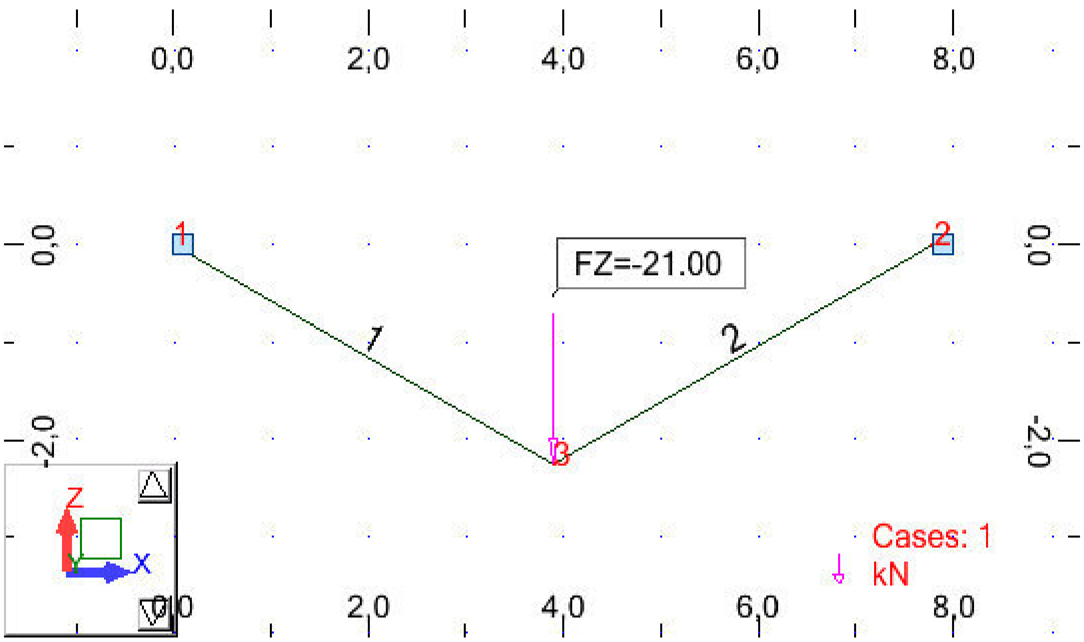
Exact agreement of results.
(taking into account different sign convention).

VERIFICATION EXAMPLE

Plane truss with nodal loads - SSLL09/89

Name of the test: SSLL09/89
Reference: AFNOR
Specification: Truss made of two bars; material: elastic, linear, isotropic.

GEOMETRY:



DATA FILE: SSLL09.str

COMPARISON:

Node	Compared result	Robot	AFNOR	Difference %
3	Displacement UZ (m)	-3.000 e-3	-3.000 e-3	0.0
1 - 3	Tensile force (N)	21.000 e+3	21.000 e+3	0.0
2 - 3	Displacement UZ (m)	21.000 e+3	21.000 e+3	0.0

CONCLUSION:

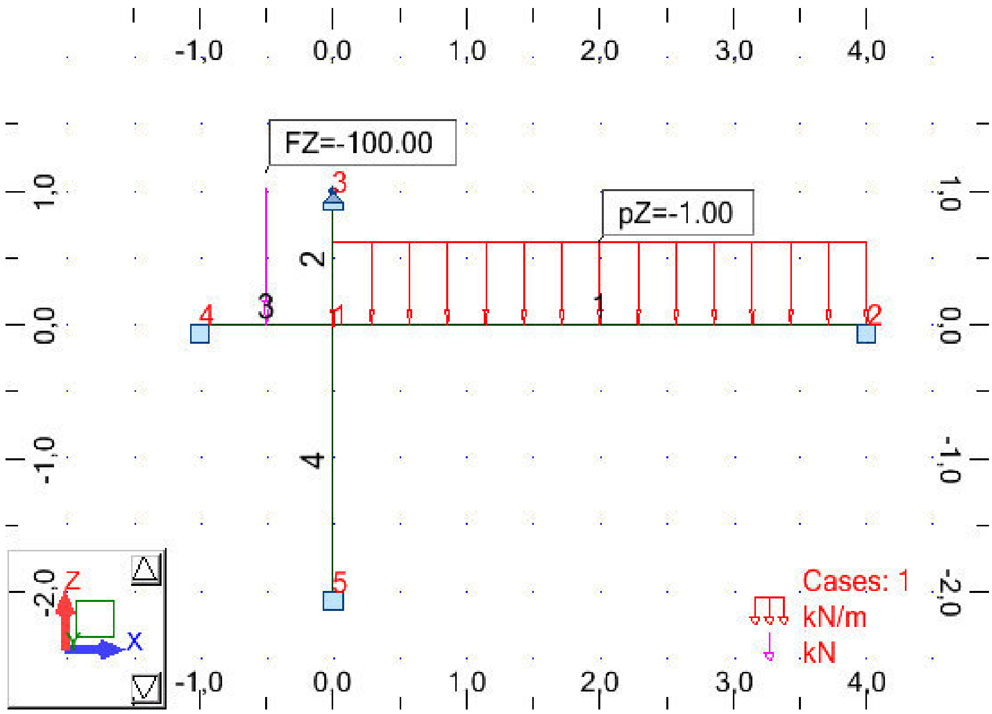
Exact agreement of results.

VERIFICATION EXAMPLE

Plane frame with uniform loads - SSLL10/89

Name of the test: SSLL10/89
Reference: AFNOR
Specification: Frame made of four bars with different moments of inertia; material: elastic, linear, isotropic (non-compressible bars assumed)

GEOMETRY:



DATA FILE: SSLL10.str

COMPARISON:

Node	Compared result	Robot	AFNOR	Difference %
1	Rotation RY (rad)	-0.227119	0.227118	0.0
1	Moment MY (Nm)	-11023.73	11023.72	0.0
2	Moment MY (Nm)	-113.559	113.559	0.0
3	Moment MY (Nm)	-12348.59	-12348.588	0.0
4	Moment MY (Nm)	-1211.2997	1211.2994	0.0

CONCLUSION:

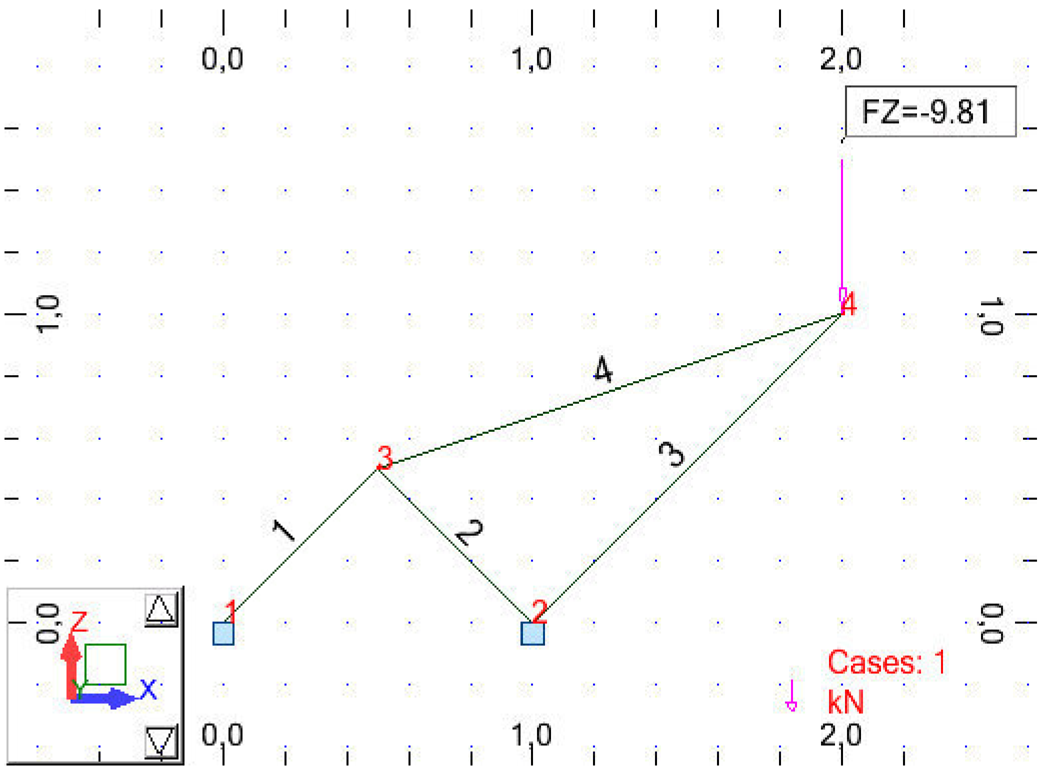
Exact agreement of results (*taking into account different sign convention*).

VERIFICATION EXAMPLE

Plane truss with nodal loads - SSLL11/89

Name of the test: SSLL11/89
Reference: AFNOR
Specification: Truss made of four bars of different cross sections; material: elastic, linear, isotropic (non-compressible bars assumed).

GEOMETRY:



DATA FILE: SSLL11.str

COMPARISON:

Node	Compared result	Robot	AFNOR	Difference %
3	Displacement UX (m)	0.26517e-3	0.26517e-3	0.0
3	Displacement UZ (m)	0.08839e-3	0.08839e-3	0.0
4	Displacement UX (m)	3.47902e-3	3.47902e-3	0.0
4	Displacement UZ (m)	-5.60084e-3	-5.60084e-3	0.0

CONCLUSION:

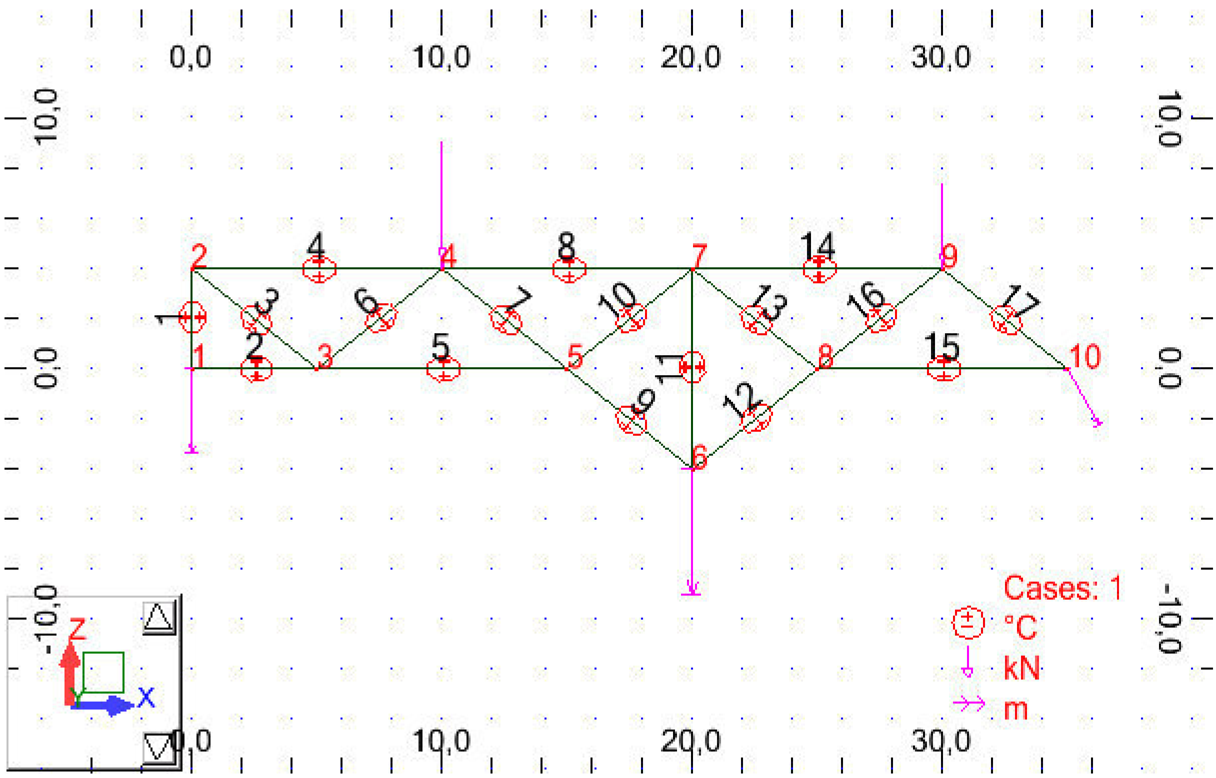
Exact agreement of results.

VERIFICATION EXAMPLE

Plane truss under thermal and displacement loadings - SSLL12/89

Name of the test: SSLL12/89
Reference: AFNOR
Specification: Plane truss - initial displacements - dilatation effect - pinned supports.

GEOMETRY:



DATA FILE: SSLL12.str

COMPARISON:

Node	Compared result	Robot	AFNOR	Difference %
6 - 8	Tension force (N)	43633	43633	0.0
8	Displacement UZ (m)	-0.01618	-0.01618	0.0

CONCLUSION:

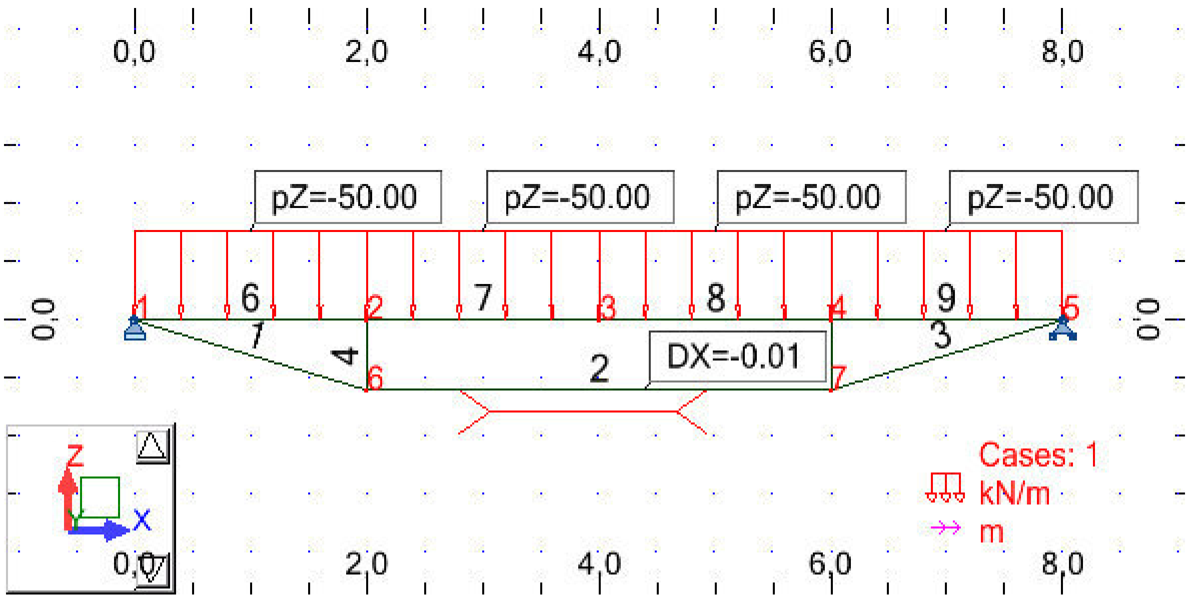
Exact agreement of results.

VERIFICATION EXAMPLE

Shortening of a tie-beam - SSLL13/89

Name of the test: SSLL13/89
Reference: AFNOR
Specification: Simply supported beam with truss elements (hinged joints), shortening of bars; elastic, linear, isotropic material.

GEOMETRY:



DATA FILE: SSLL13.str

COMPARISON:

Node	Compared result	Robot	AFNOR	Difference %
6 - 7	Tension force (N)	584584	584584	0.0
3	Moment MY (Nm)	49249.5	49249.5	0.0
2	Displacement UZ (m)	-0.5428 e-3	-0.5428 e-3	0.0

CONCLUSION:

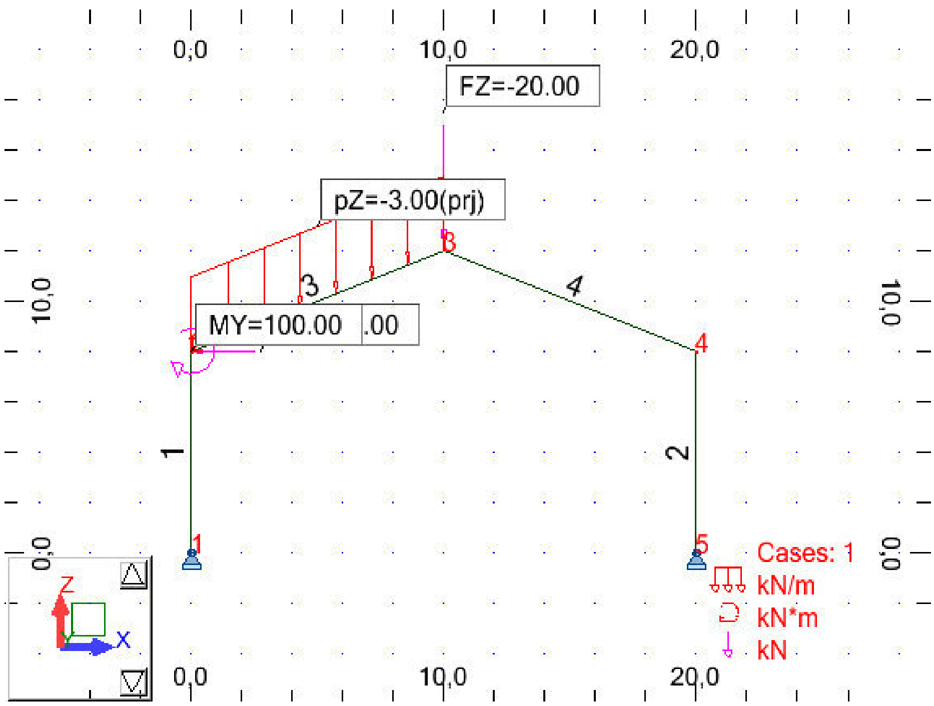
Exact agreement of results.

VERIFICATION EXAMPLE

Plane frame bending - SSLL14/89

Name of the test: SSLL14/89
Reference: AFNOR
Specification: Simply supported symmetrical frame with asymmetric load; material: elastic, linear, isotropic (non-compressible bars assumed).

GEOMETRY:



DATA FILE: SSLL14.str

COMPARISON:

Node	Compared result	Robot	AFNOR	Difference %
1	Vertical reaction (N)	31500.0	31500.0	0.0
1	Horizontal reaction (N)	20239.4	20239.4	0.0
3	Displacement UZ (m)	-0.03072	-0.03072	0.0

CONCLUSION:

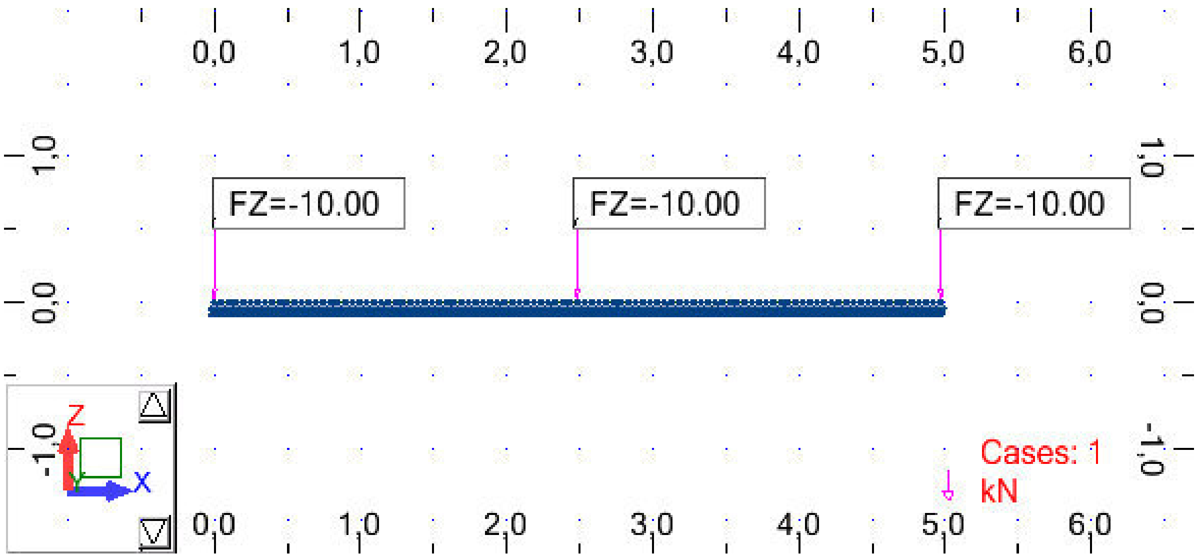
Exact agreement of results.

VERIFICATION EXAMPLE

Beam on elastic (Winkler's) soil foundation - SSLL15/89

Name of the test: SSLL15/89
Reference: AFNOR
Specification: Simple beam on bidirectional, elastic foundation (Winkler's foundation) elastic, linear, isotropic material

GEOMETRY:



DATA FILE: SSLL15.str (dense division on beam-elements with elastic supports), SSLL15R.rtd (2 Winkler's beam-elements, without nodal supports)

COMPARISON:

Node	Compared result	Robot (SSLL15.str)	AFNOR	Difference %
51	Moment MY (Nm)	-5758	5759	0.017
51	Displacement UZ (m)	-0.006843	-0.006844	0.015
1	Displacement UZ (m)	-0.007859	-0.007854	0.064
1	Rotation RY (rad)	-0.000706	-0.000706	0.0

Node	Compared result	Robot (SSLL15R.rtd)	AFNOR	Difference %
2	Moment MY (Nm)	-5759	5759	0.0
2	Displacement UZ (m)	-0.0068434	-0.006844	0.009
1	Displacement UZ (m)	-0.0078588	-0.007854	0.061
1	Rotation RY (rad)	-0.000706	-0.000706	0.0

CONCLUSIONS:

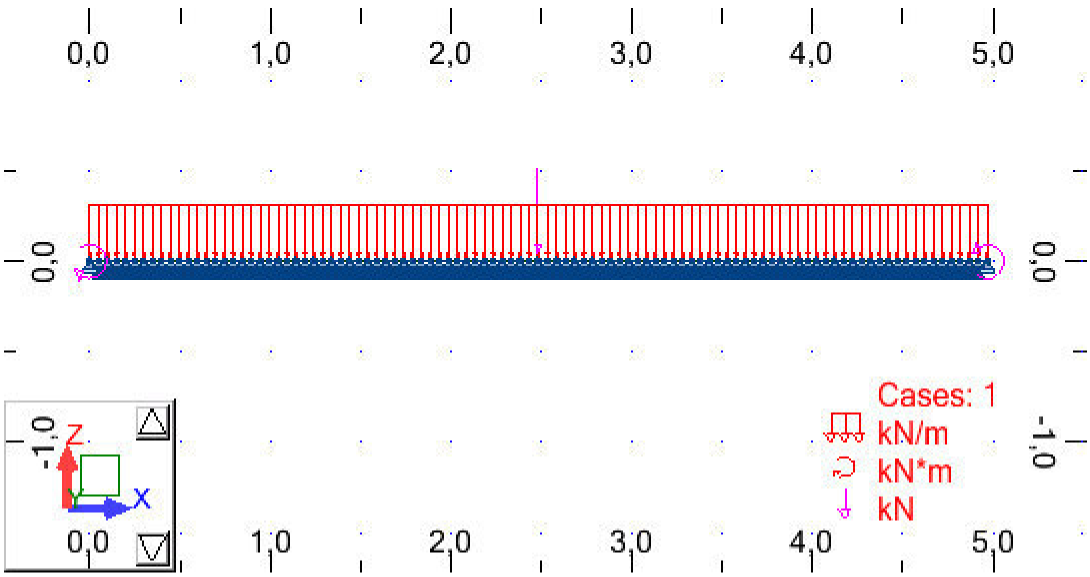
Excellent agreement of results.
(taking into account different sign convention).

VERIFICATION EXAMPLE

Beam on elastic (Winkler's) soil foundation - SSLL16/89

Name of the test: SSLL16/89
Reference: AFNOR
Specification: Simple beam on bidirectional, elastic foundation (Winkler's foundation) elastic, linear, isotropic material

GEOMETRY:



DATA FILE: SSLL16.str (dense division on beam-elements with elastic supports), SSLL16R.rtd (2 Winkler's beam-elements, without nodal supports)

COMPARISON:

Node	Compared result	Robot (SSLL16.str)	AFNOR	Difference %
1	Rotation RY (rad)	-0.003045	-0.003045	0.0
1	Vertical reaction FZ (N)	11675	11674	0.008
51	Displacement UZ (m)	-0.00423099	-0.00423326	0.054
51	Moment MY (Nm)	-33839	-33840	0.003

Node	Compared result	Robot (SSLL16R.rtd)	AFNOR	Difference %
1	Rotation RY (rad)	-0.003045	-0.003045	0.0
1	Vertical reaction FZ (N)	11674	11674	0.0
2	Displacement UZ (m)	-0.00423299	-0.00423326	0.006
2	Moment MY (Nm)	-33840	-33840	0.0

CONCLUSIONS:

Excellent agreement of results.
(taking into account different sign convention).

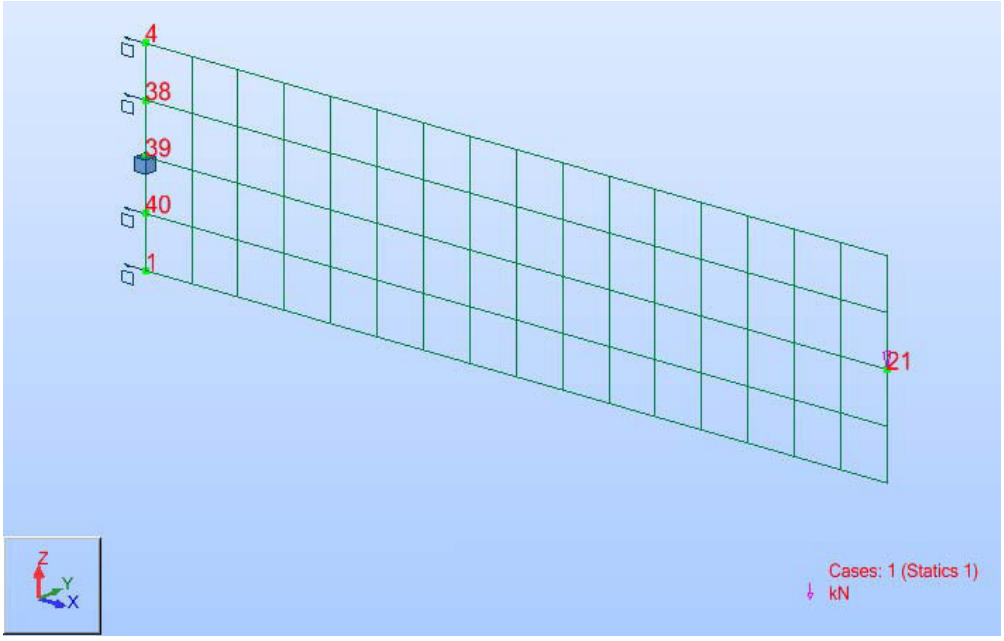
2. PLATE/SHELL STRUCTURES

VERIFICATION EXAMPLE

Rectangular membrane under in-plane shear - SSLP01/89

Name of the test: SSLP01/89
Reference: AFNOR
Specification: Rectangular shell: in-plane bending and shear.

GEOMETRY: Mesh 4x16 (3mm size square FE), point load in node 21



DATA FILE SSLP01.str

COMPARISON:

Node	Compared result	Robot	AFNOR	Difference %
21	Displacement UZ (mm)	0.3582	0.3573	0.252
4	Stress (N/mm2)	-79.56	-80.0	0.550

CONCLUSION:

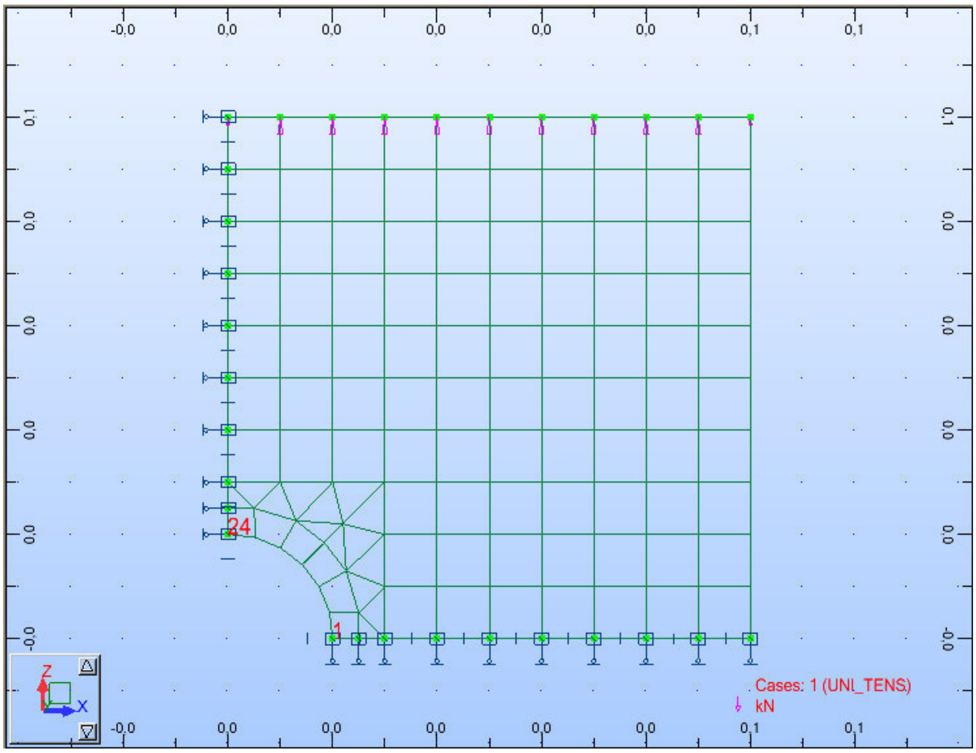
Very good agreement of results.

VERIFICATION EXAMPLE

Tension of perforated membrane - SSLP02/89

Name of the test: SSLP02/89
Reference: AFNOR
Specification: Simple tension of perforated membrane.

GEOMETRY: ¼ of a model analyzed (due to symmetry) with a mesh 10x10



DATA FILE SSLP02.str

COMPARISON:

Node	Compared result	Robot	AFNOR	Difference %
1	Stress $\sigma_{\theta\theta}$ (N/mm2)	- 7.799	- 7.5	3.987
24	Stress $\sigma_{\theta\theta}$ (N/mm2)	- 2.766	- 2.5	10.640

CONCLUSION:

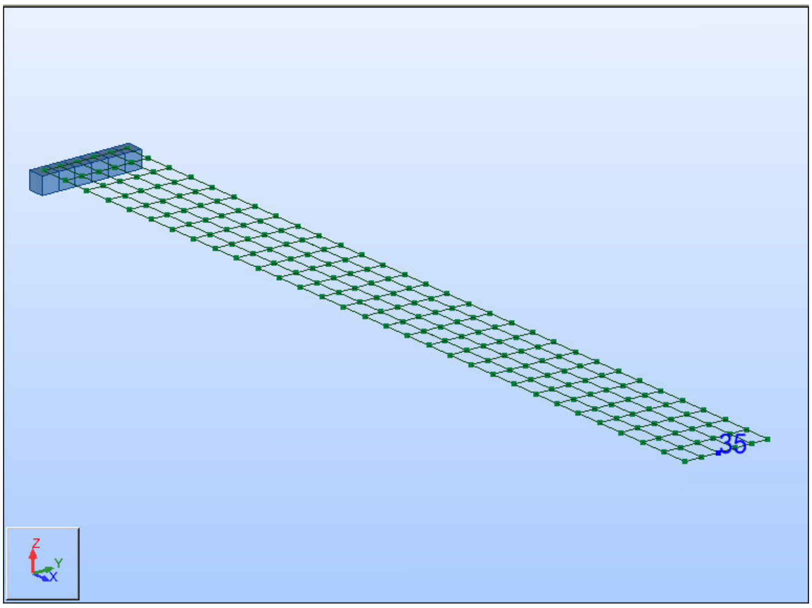
Poor agreement of results.

VERIFICATION EXAMPLE

Rectangular plate: cantilever slab - SSLS01/89

Name of the test: SSLS01/89
Reference: AFNOR
Specification: Cantilever slab under uniform pressure

GEOMETRY:



DATA FILE SSLS01.str

COMPARISON:

Node	Compared result	Robot	AFNOR	Difference %
35	Displacement UZ (mm)	- 95.92	- 95.90	0.021

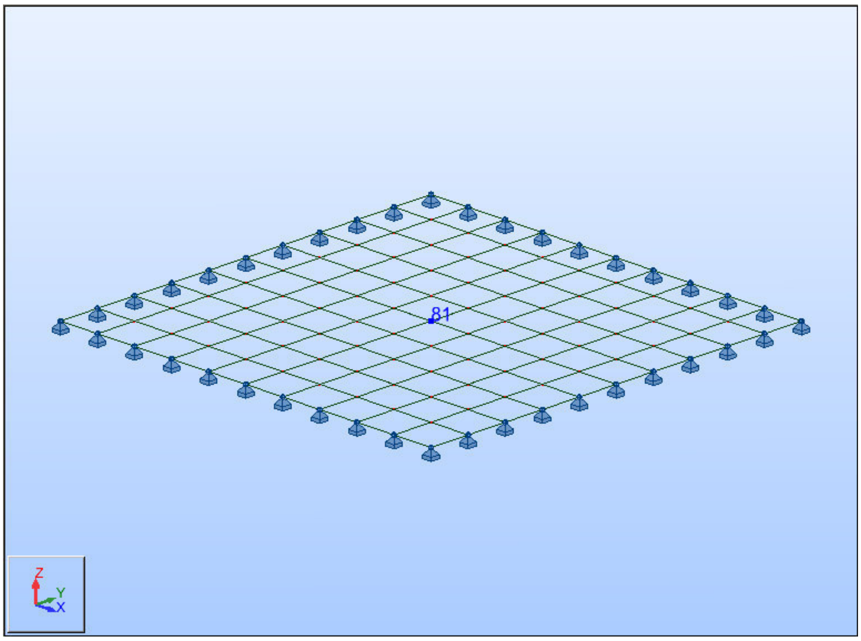
CONCLUSION:
Excellent agreement of results.

VERIFICATION EXAMPLE

Simply supported square plate - SSLS02/89

Name of the test: SSLS02/89
Reference: AFNOR
Specification: Simply supported square plate under self weight.

GEOMETRY:



DATA FILE: SSLS02.str

COMPARISON:

Node	Compared result	Robot	AFNOR	Difference %
81	Displacement UZ (mm)	- 16.47	- 16.45*	0.122

* *"Guide..." presents an incorrect value (compare with SSLS 24/89)*

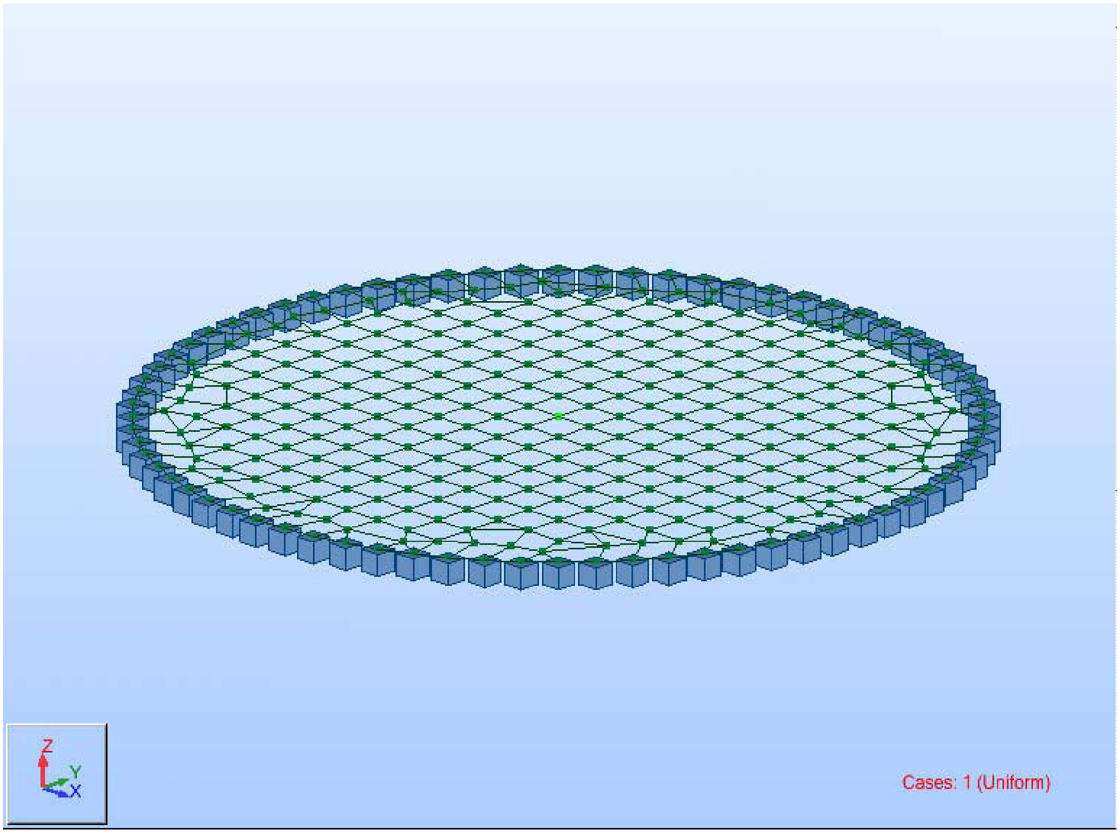
CONCLUSION:
Excellent agreement of results.

VERIFICATION EXAMPLE

Circular plate under uniform load - SSLS03/89

Name of the test: SSLS03/89
Reference: AFNOR
Specification: Circular plate with clamped edges under uniform load

GEOMETRY:



DATA FILES: SSLS03.str

COMPARISON:

Node	Compared result	Robot	AFNOR	Difference %
1	Displacement UZ (mm)	- 6.477	- 6.500	0.354

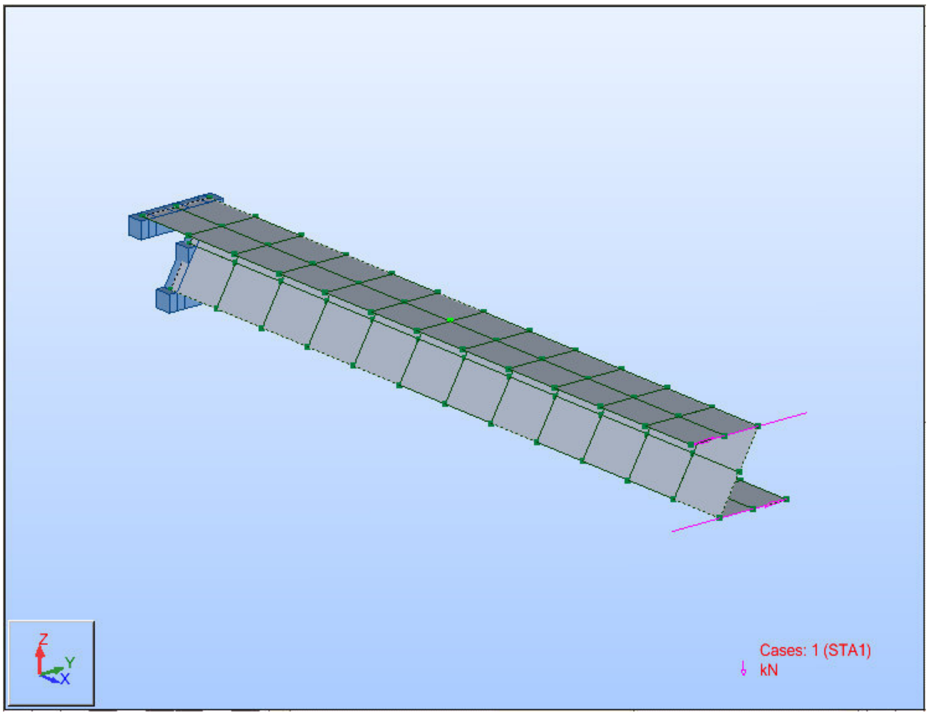
CONCLUSION:
Very good agreement of results.

VERIFICATION EXAMPLE

Beam of Z-section (using shell elements) - SSLS04/89

Name of the test: SSLS04/89
Reference: AFNOR

GEOMETRY:



DATA FILE: SSLS04.str

COMPARISON:

Node	Compared result	Robot	AFNOR	Difference %
117	Displacement UY (mm)	- 6.946	- 7.150	2.853

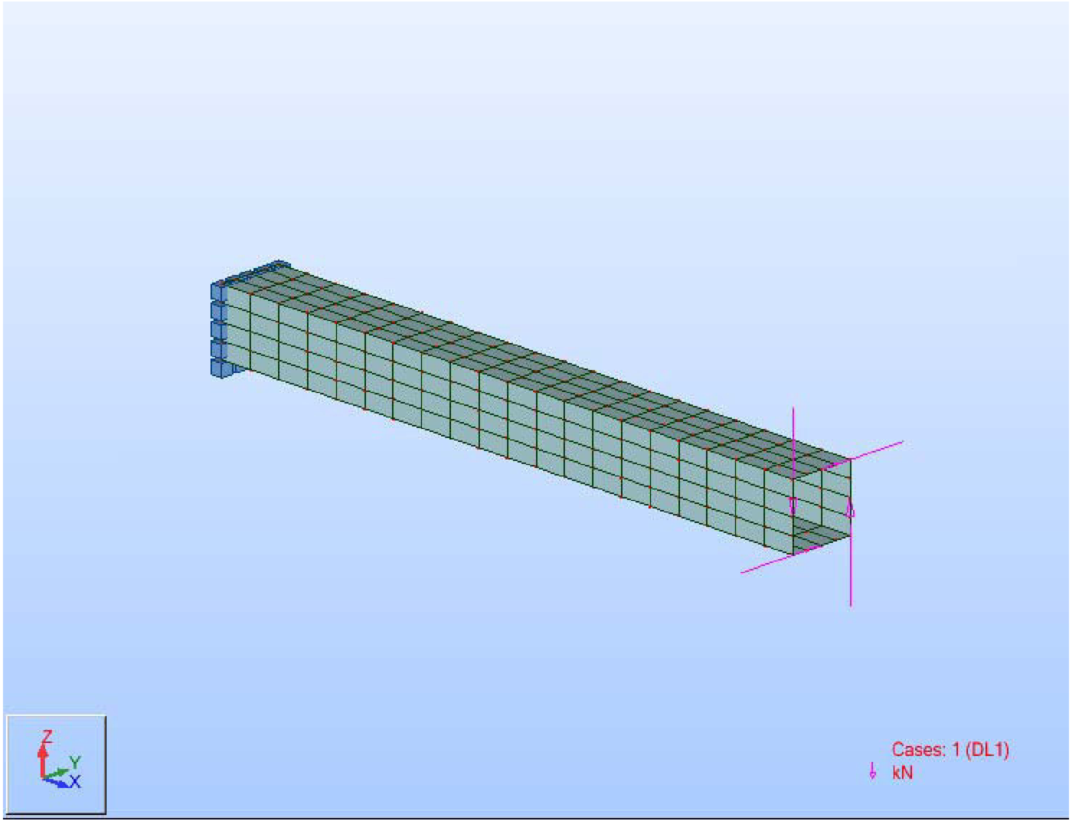
CONCLUSION:
Good agreement of results.

VERIFICATION EXAMPLE

Box section in torsion (using shell elements) - SSLS05/89

Name of the test: SSLS05/89
Reference: AFNOR
Specification: Shell - Box section - Shear - Torsion.

GEOMETRY:



DATA FILE: SSLS05.str

COMPARISON:

Node	Compared result	Robot	AFNOR	Difference %
158	Displacement UY (m)	- 0.616 e-3	- 0.617 e-3	0.162
158	Rotation RX (rad)	0.0123 e-3	0.0123 e-3	0.0
83	Displacement UZ (m)	- 0.986 e-3	- 0.987 e-3	0.101
83	Rotation RX (rad)	0.0197 e-3	0.0197 e-3	0.0

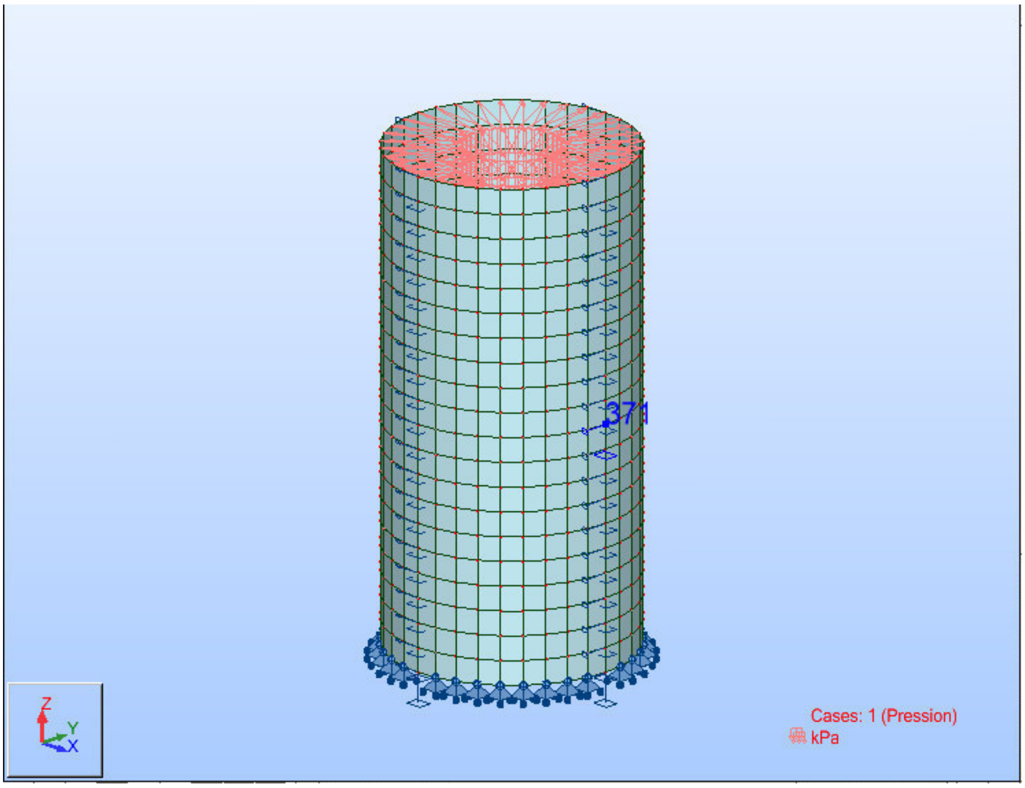
CONCLUSION:
Excellent agreement of results.

VERIFICATION EXAMPLE

Thin-walled cylinder under uniform radial pressure - SSLS06/89

Name of the test: SSLS06/89
Reference: AFNOR
Specification: Shell - Cylinder - Material: elastic - Pressure

GEOMETRY:



DATA FILE: SSLS06.str

COMPARISON:

Node	Compared result	Robot	AFNOR	Difference %
371	Displacement UX (mm)	2.371 e-3	2.380 e-3	0.378
371	Radial stress (kPa)	498.1	500.0	0.380
741	Displacement UZ (mm)	- 2.964 e-3	- 2.860 e-3	3.636

CONCLUSION:

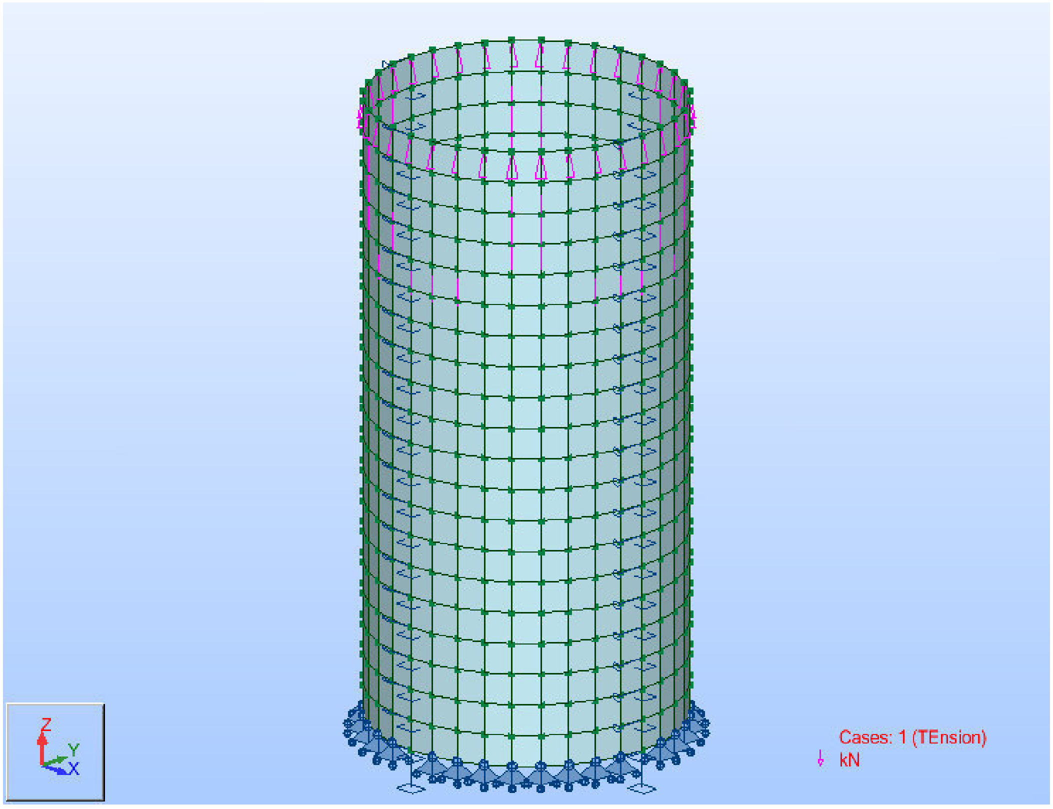
Good agreement of results.

VERIFICATION EXAMPLE

Thin-walled cylinder with uniform axial load - SSLS07/89

Name of the test: SSLS07/89
Reference: AFNOR
Specification: Shell - Material: elastic - uniform load - Cylinder

GEOMETRY:



DATA FILE: SSLS07.str

COMPARISON:

Node	Compared result	Robot	AFNOR	Difference %
371	Displacement UX (mm)	- 7.152 e-3	-7.140 e-3	0.168
371	Tension stress (kPa)	500.6	500.0	0.120
741	Displacement UZ (mm)	9.626 e-3	9.520 e-3	1.113

CONCLUSION:

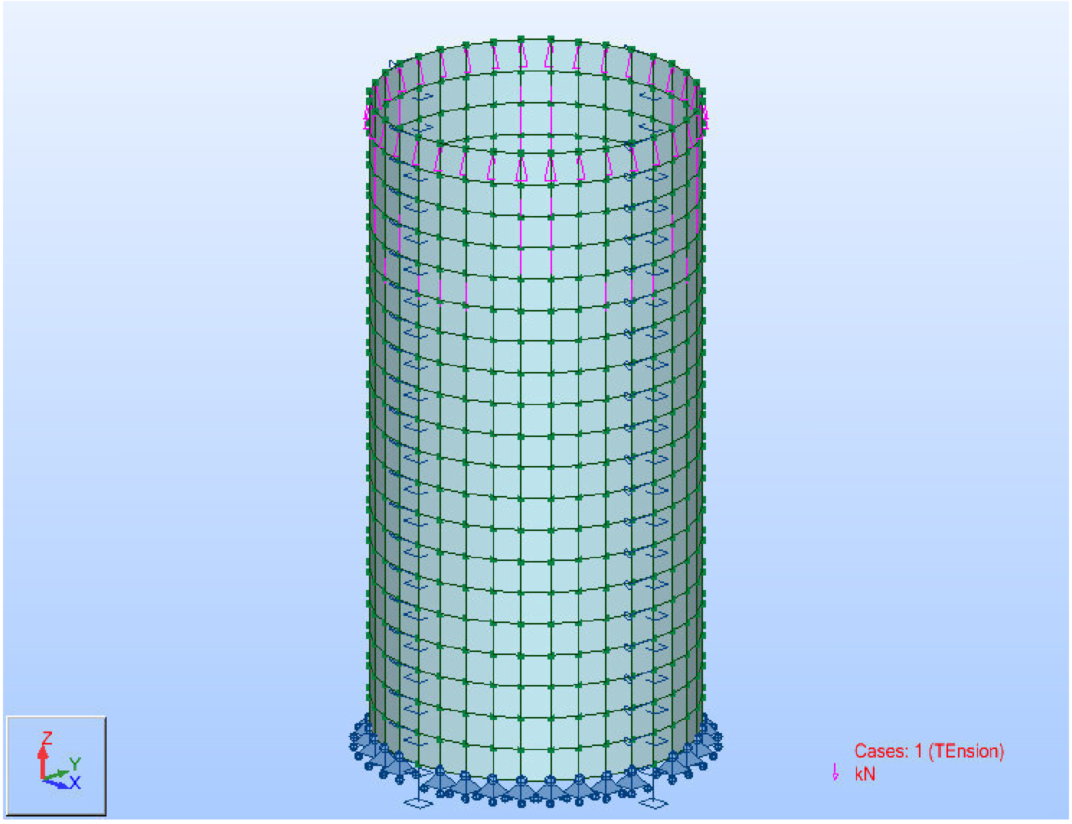
Very good agreement of results.

VERIFICATION EXAMPLE

Thin-walled cylinder under hydrostatic pressure - SSLS08/89

Name of the test: SSLS08/89
Reference: AFNOR
Specification: Shell - Material: elastic - Hydrostatic pressure - Cylinder

GEOMETRY:



DATA FILE: SSLS08.str

COMPARISON:

Node	Compared result	Robot	AFNOR	Difference %
371	Displacement UX (mm)	2.371 e-3	2.380 e-3	0.379
371	Radial stress (kPa)	498.1	500.0	0.380
741	Displacement UZ (mm)	-2.964 e-3	-2.860 e-3	0.489

CONCLUSION:

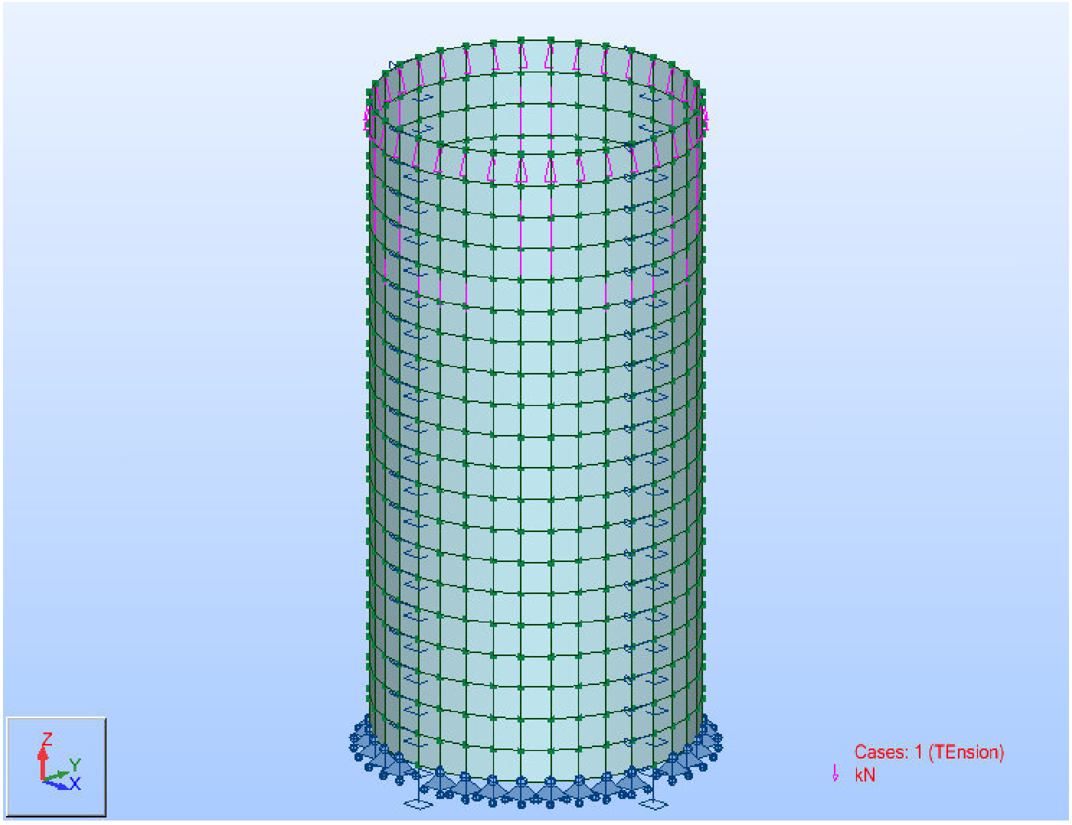
Excellent agreement of results.

VERIFICATION EXAMPLE

Thin-walled cylinder under self-weight - SSLS09/89

Name of the test: SSLS09/89
Reference: AFNOR
Specification: Shell - Material: elastic – Self-weight - Cylinder

GEOMETRY:



DATA FILE: SSLS09.str

COMPARISON:

Node	Compared result	Robot	AFNOR	Difference %
371	Displacement UX (mm)	0.2243 e-3	0.2245 e-3	0.089
38	Radial stress (kPa)	312.5	314.2	0.637
741	Displacement UZ (mm)	-0.3019 e-3	-0.2990 e-3	0.956

CONCLUSION:

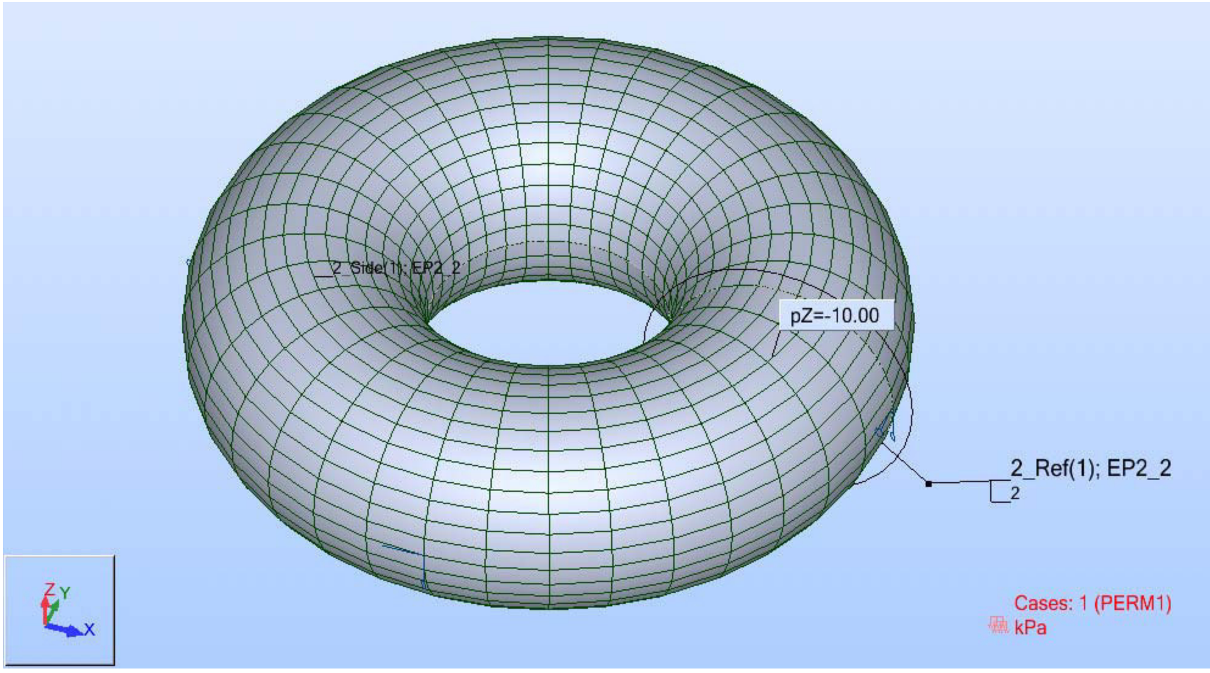
Excellent agreement of results.

VERIFICATION EXAMPLE

Torus under uniform internal pressure - SSLS10/89

Name of the test: SSLS10/89
Reference: AFNOR
Codification: Shell - Torus - Material: elastic - Pressure.

GEOMETRY:



DATA FILE: SSLS10.str

COMPARISON:

Node	Compared result	Robot	AFNOR	Difference %
335 (internal)	Displacement UY (m)	0.789 e-7	1.19 e-7	33.7
	Horizontal stress σ_{XX} (Pa)	2.39 e+5	2.50 e+5	4.40
	Vertical stress σ_{YY} (Pa)	7.36 e+5	7.50 e+5	1.87
362 (external)	Displacement UY (m)	1.93 e-7	1.79 e-7	7.25
	Horizontal stress σ_{XX} (Pa)	2.60 e+5	2.50 e+5	4.00
	Vertical stress σ_{YY} (Pa)	4.16 e+5	4.17 e+5	0.24

CONCLUSION:

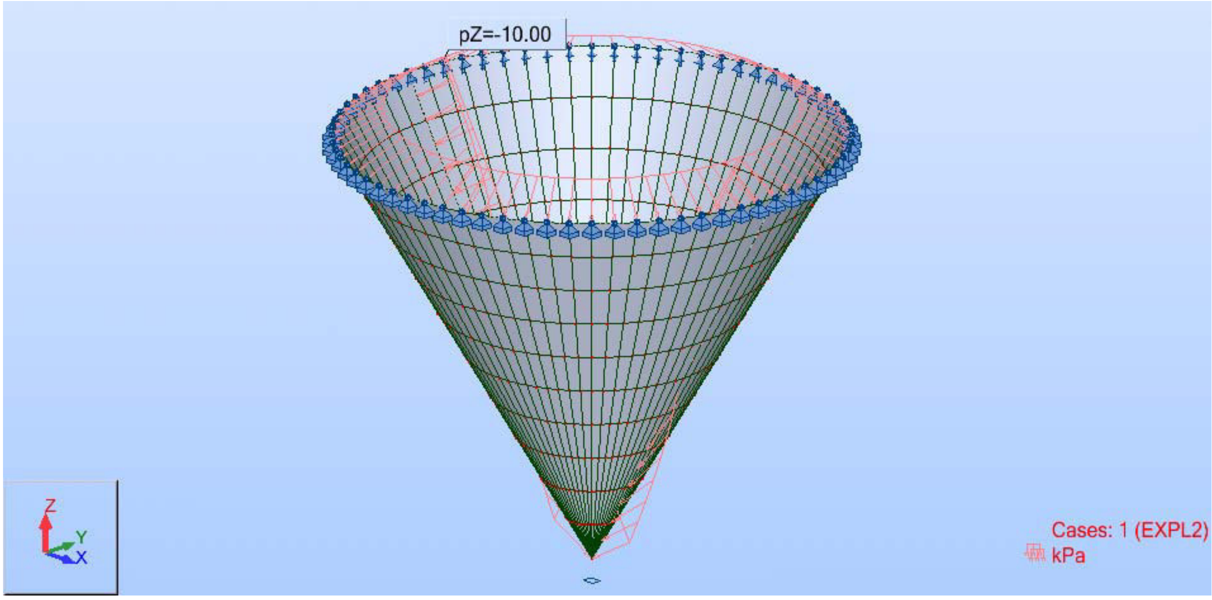
Good agreement of results.

VERIFICATION EXAMPLE

Thin-walled cone subjected to uniform internal pressure - SSLS11/89

Name of the test: SSLS11/89
Reference: AFNOR
Specification: Shell - Cone - Material: elastic - Pressure.

GEOMETRY:



DATA FILE: SSLS11.rtd

COMPARISON:

Node	Compared result	Robot	AFNOR	Difference %
7 (mid-height)	Vertical stress (Pa)	1.45 e+5	1.44 e+5	0.69
	Horizontal stress (Pa)	2.88 e+5	2.89 e+5	0.35
	Displacement UX (δ_R) (m)	0.5841 e-6	0.5842 e-6	0.02

CONCLUSION:

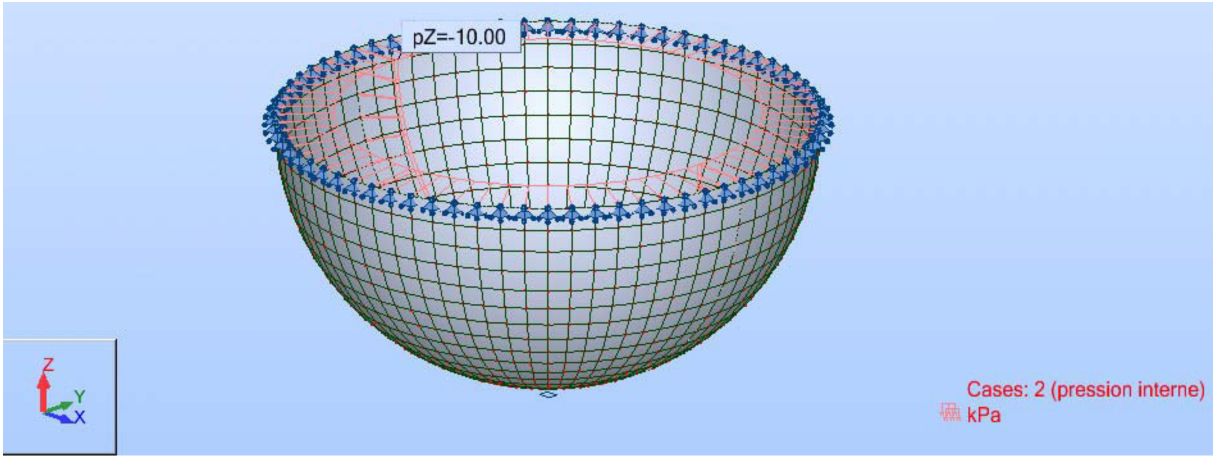
Excellent agreement of results.

VERIFICATION EXAMPLE

Spherical shell subjected to a pressure - SSLS14/89

Name of the test: SSLS14/89
Reference: AFNOR
Codification: Shell - spherical cup - Material: elastic - Uniform pressure

GEOMETRY:



DATA FILE: SSLS14.rtd

COMPARISON:

Node	Compared result	Robot	AFNOR	Difference %
All	Horizontal stress (Pa)	2.51 e+5	2.50 e+5	0.4
1	Displacement δ_R (m)	8.33 e-7	8.33 e-7	0.0

CONCLUSION:

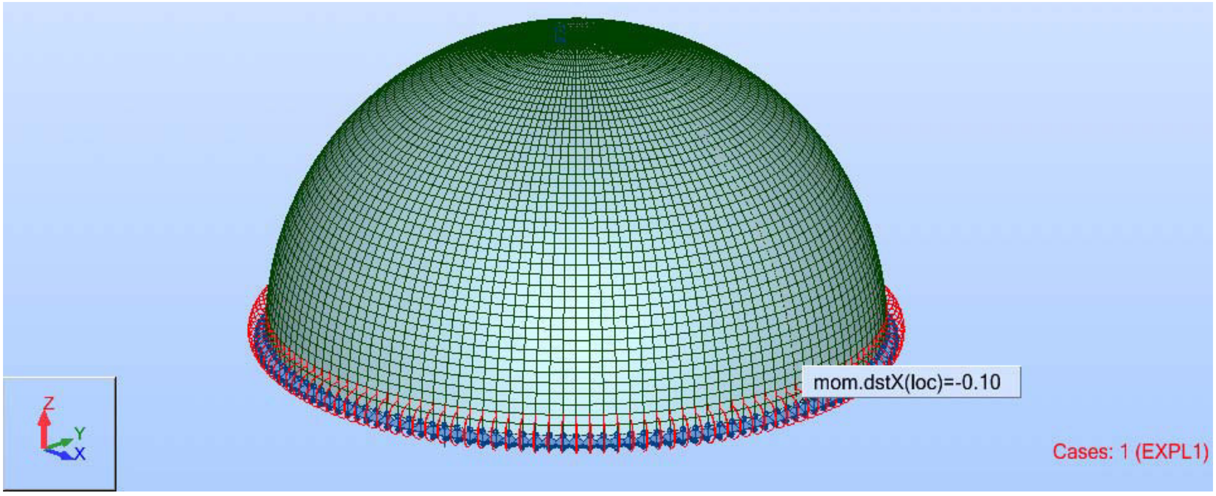
Excellent agreement of results.

VERIFICATION PROBLEM

Spherical shell subjected to a moment - SSLS16/89

Name of the test: SSLS16/89
Reference: AFNOR
Codification: Shell - spherical cup - Material: elastic - Uniform moment

GEOMETRY:



DATA FILE: SSLS16.rtd

COMPARISON:

Node	Compared result	Robot	AFNOR	Difference %
692	Horizontal stress (Pa)	8.34 e+5	8.26 e+5	0.96
	Displacement δ_R (m)	3.93 e-6	3.93 e-6	0.0

CONCLUSION:

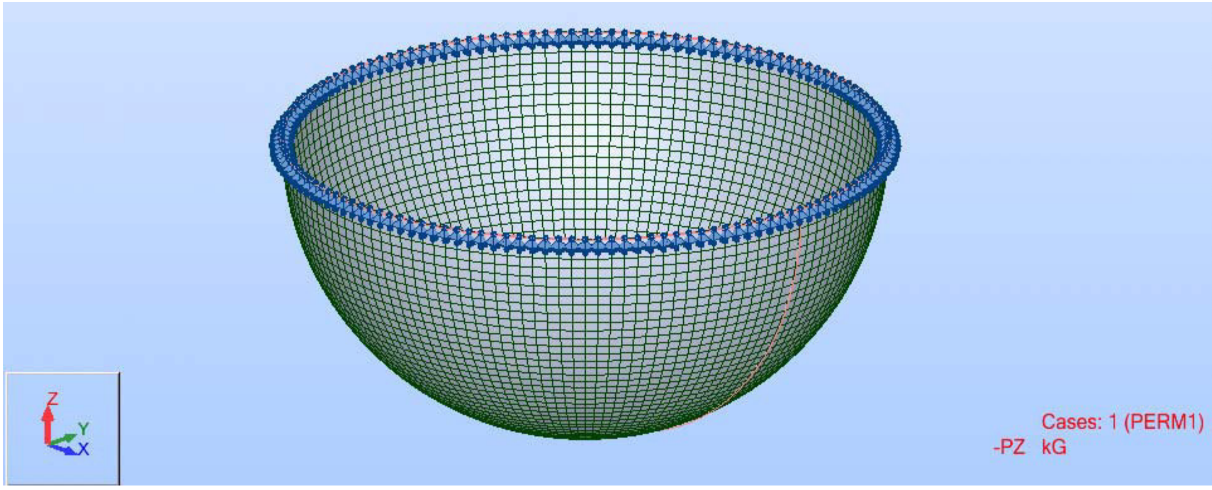
Excellent agreement of results.

VERIFICATION EXAMPLE

Spherical shell - SSLS17/89

Name of the test: SSLS17/89
Reference: AFNOR
Codification: Shell - spherical cup - Material: elastic – Self weight

GEOMETRY:



DATA FILE: SSLS17.rtd

COMPARISON:

Node	Compared result	Robot	AFNOR	Difference %
231	Horizontal stress (Pa)	7.59 e+4	7.85 e+4	3.31
	Vertical stress (Pa)	- 8.19 e+4	- 7.85 e+4	4.33
	Displacement δ_R (m)	4.90 e-7	4.86 e-7	0.82

CONCLUSION:

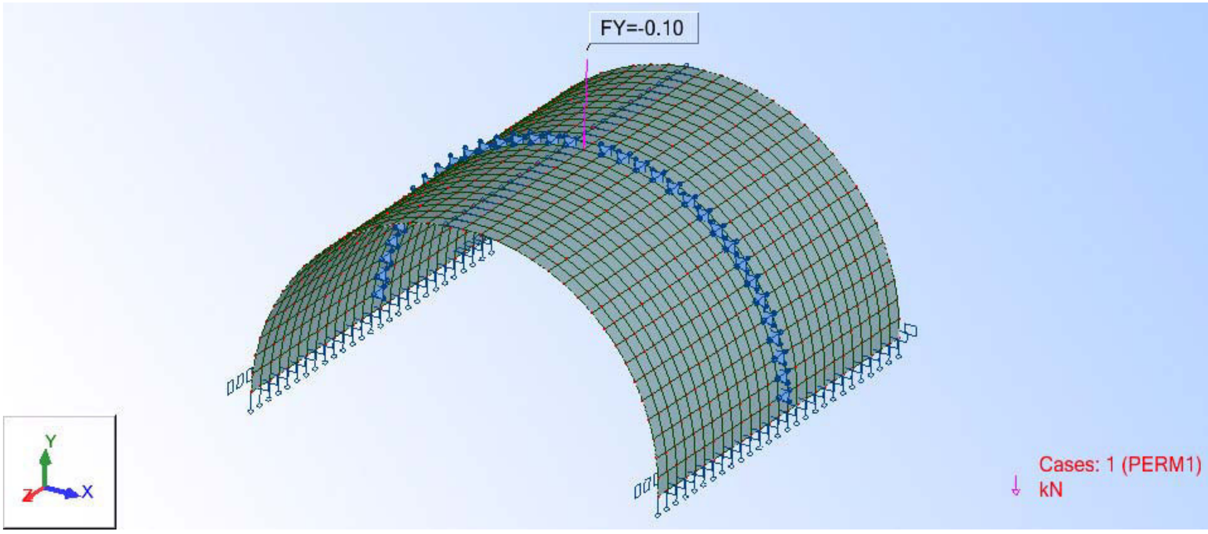
Good agreement of results.

VERIFICATION EXAMPLE

Cylindrical shell subjected to concentrated force - SSLS20/89

Name of the test: SSLS20/89
Reference: AFNOR
Codification: Cylindrical shell - Material: elastic - Concentrated forces.

GEOMETRY:



DATA FILE: SSLS20.rtd

COMPARISON:

Node	Compared result	Robot	AFNOR	Difference %
500	Displacement UY (m)	-11.374 e-2	-11.390 e-2	0.14

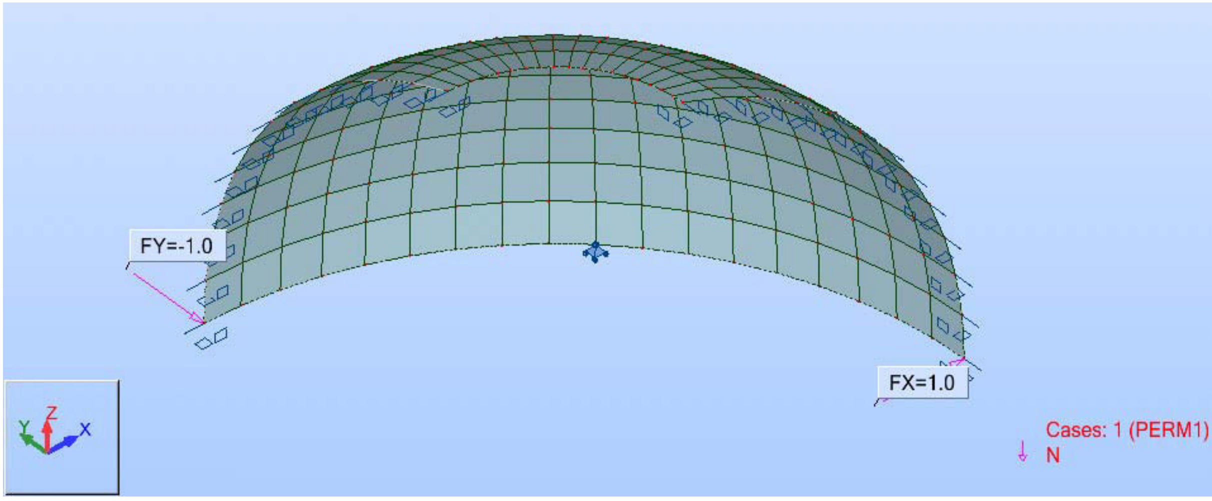
CONCLUSION:
Excellent agreement of results.

VERIFICATION EXAMPLE

Spherical shell with an opening - SSLS21/89

Name of the test: SSLS21/89
Reference: AFNOR
Codification: Spherical shell - Material: elastic - Concentrated forces.

GEOMETRY:



DATA FILE: SSLS21.rtd

COMPARISON:

Node	Compared result	Robot	AFNOR	Difference %
1	Displacement UX (m)	101.47 e-3	94.00 e-3	7.95

CONCLUSION:

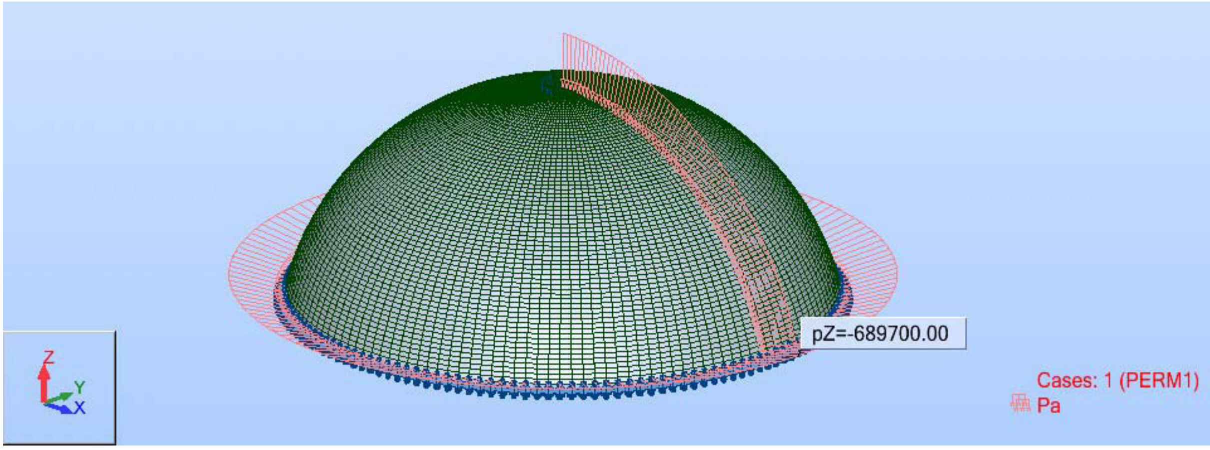
Results correct.

VERIFICATION EXAMPLE

Spherical dome subjected to uniform external pressure - SSLS22/89

Name of the test: SSLS22/89
Reference: AFNOR
Specification: Spherical shell - Material: elastic - pressure.

GEOMETRY:



DATA FILE: SSLS22.rtd

COMPARISON:

Node	Compared result	Robot	AFNOR	Difference %
5794	Displacement UX (m)	1.74 e-3	1.73 e-3	0.58
5824	Displacement UX (m)	1.02 e-3	1.02 e-3	0.0
5794	Vertical stress σ_{YY} (Pa)	-0.68 e+8	-0.74 e+8	8.11
5824	Vertical stress σ_{YY} (Pa)	-0.69 e+8	-0.68 e+8	1.47

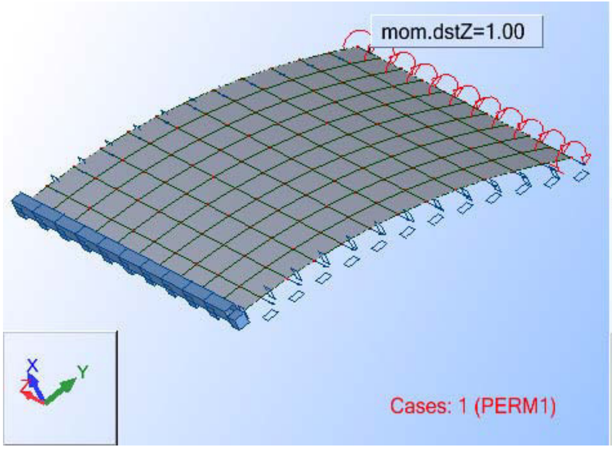
CONCLUSION:
Results correct.

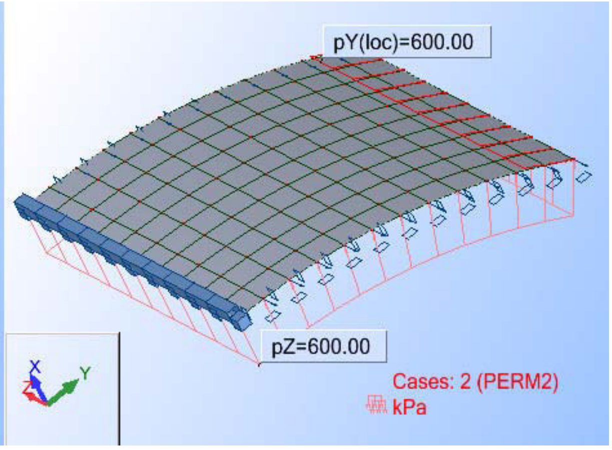
VERIFICATION EXAMPLE

Cylindrical membrane subjected to bending - SSLS23/89

Name of the test: SSLS23/89
Reference: AFNOR
Specification: Bending - Membrane effect

GEOMETRY:





DATA FILE: SSLS23.rtd

COMPARISON:

Case	Node	Compared result	Robot	AFNOR	Difference %
1 (flexion)	87	Stress σ_{XX} (MPa)	60.00	60.00	0.00
2 (membrane)	87	Stress σ_{XX} (MPa)	59.99	60.00	0.02

CONCLUSION:

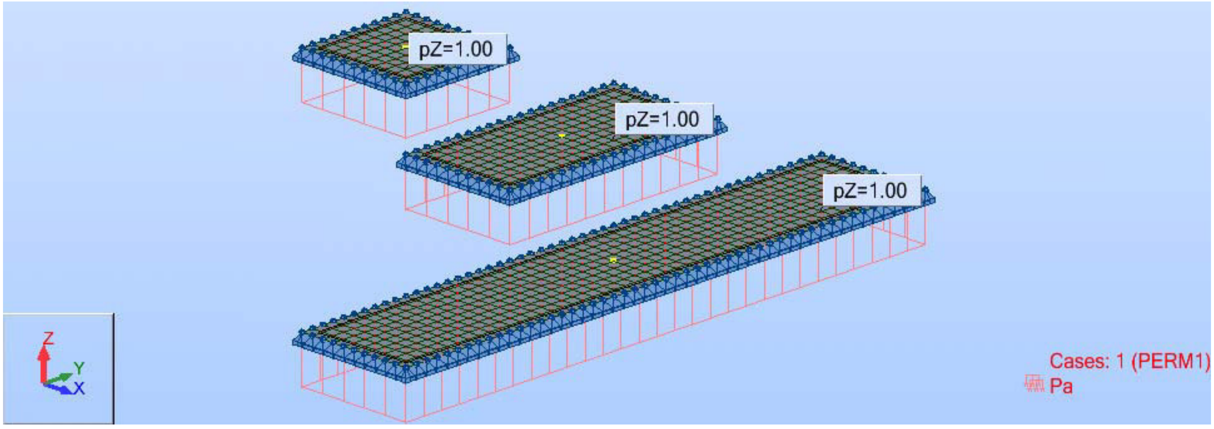
Excellent agreement of results.

VERIFICATION EXAMPLE

Simply supported rectangular plate with uniform load - SSLS24/89

Name of the test: SSLS24/89
Reference: AFNOR
Specification: Plate - Pressure - Simple support.

GEOMETRY:



DATA FILE: SSLS24.rtd

COMPARISON:

Case where b/a=1:

Node	Compared result	Robot	AFNOR	Difference %
81	Deflection (m)	44.35	44.30	0.10
81	Moment Mxx (Nm/m)	4.82	4.79	0.63
81	Moment Myy (Nm/m)	4.82	4.79	0.63

Case where b/a=2:

Node	Compared result	Robot	AFNOR	Difference %
267	Deflection (m)	110.16	110.06	0.40
267	Moment Mxx (Nm/m)	10.20	10.17	0.32
267	Moment Myy (Nm/m)	4.63	4.64	0.19

Case where b/a=5:

Node	Compared result	Robot	AFNOR	Difference %
693	Deflection (m)	140.53	141.60	0.75
693	Moment Mxx (Nm/m)	12.46	12.46	0.03
693	Moment Myy (Nm/m)	3.77	3.75	0.63

CONCLUSION:

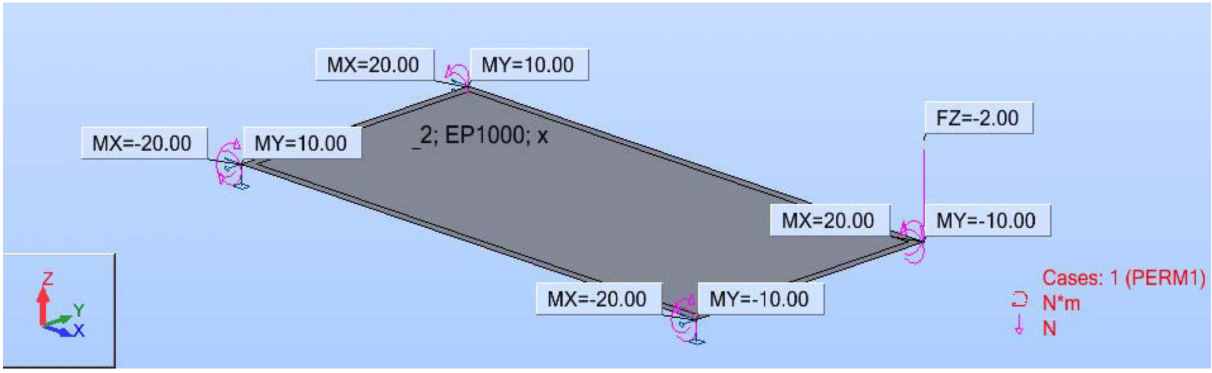
Excellent agreement of results.

VERIFICATION EXAMPLE

Simply supported rectangular plate with bending moment - SSLS26/89

Name of the test: SSLS26/89
Reference: AFNOR
Specification: Plate - Pressure - Simple support – Nodal moment

GEOMETRY:



DATA FILE: SSLS26.rtd

COMPARISON:

Node	Compared result	Robot	AFNOR	Difference %
1	Displacement UZ (m)	-12.44	-12.48	0.32

CONCLUSION:

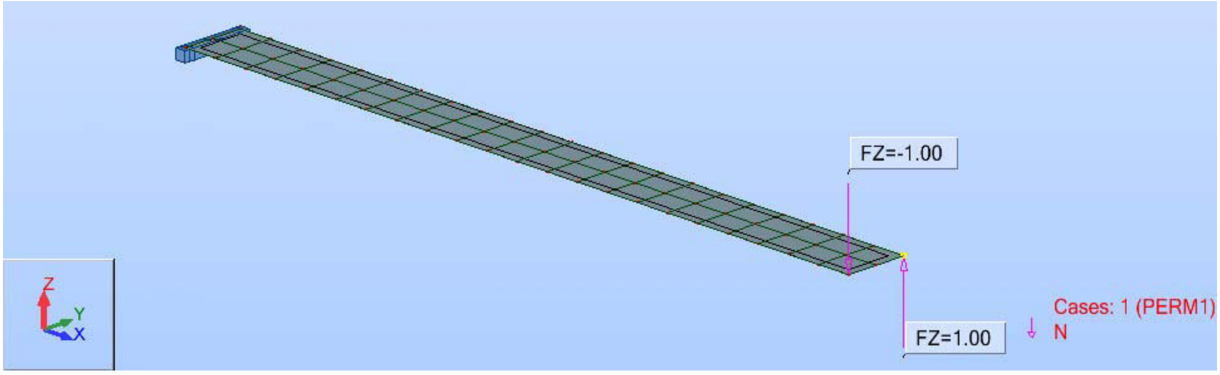
Excellent agreement of results.

VERIFICATION EXAMPLE

Plate under perpendicular shear - SSLS27/89

Name of the test: SSLS27/89
Reference: AFNOR
Specification: Plate under perpendicular shear with one edge fixed

GEOMETRY:



DATA FILE: SSLS27.rtd

COMPARISON:

Node	Compared result	Robot	AFNOR	Difference %
3	Displacement Z (m)	35.39 e-3	35.37e-3	0.06

CONCLUSION:
Excellent agreement of results.

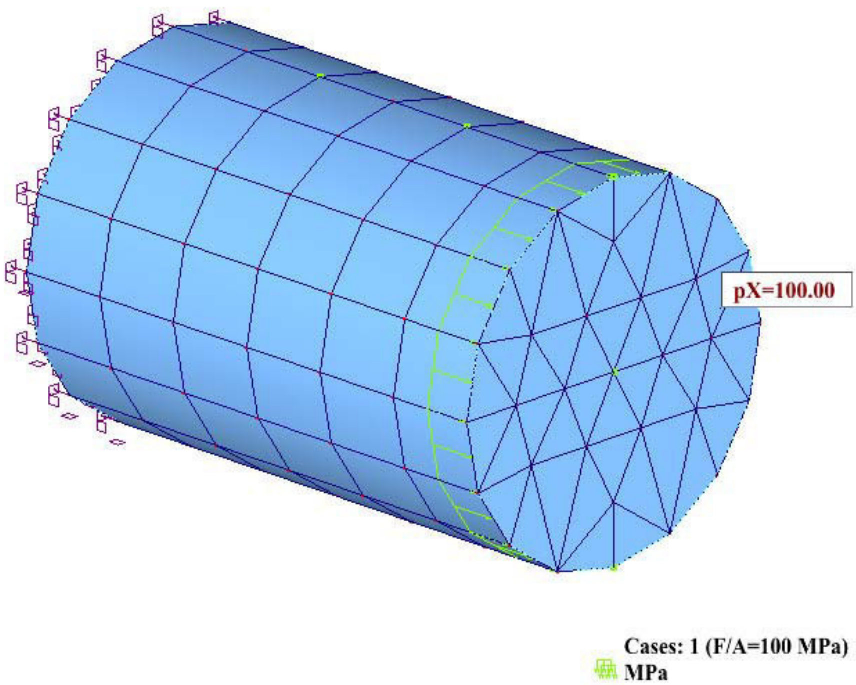
3. VOLUMIC STRUCTURES

VERIFICATION EXAMPLE

Solid cylinder subjected to simple tension - SSLV01/89

Name of the test: SSLV 01/89
Reference: AFNOR
Specification: Solid cylinder - Tension - compression - Poisson's coefficient.

GEOMETRY:



DATA FILE: SLV01.rtd

COMPARISON:

Node	Compared result	Robot	AFNOR	Difference %
200	Displacement UX (m)	1.500 e-3	1.500 e-3	0.0
214	Displacement UX (m)	1.500 e-3	1.500 e-3	0.0
208	Displacement UX (m)	1.500 e-3	1.500 e-3	0.0
138	Displacement UX (m)	1.000 e-3	1.000 e-3	0.0
76	Displacement UX (m)	0.500 e-3	0.500 e-3	0.0
200	Displacement UZ (m)	-0.1497 e-3	-0.1500 e-3	0.200
138	Displacement UZ (m)	-0. 1497 e-3	-0.1500 e-3	0.200
76	Displacement UZ (m)	-0. 1497 e-3	-0.1500 e-3	0.200

CONCLUSIONS:

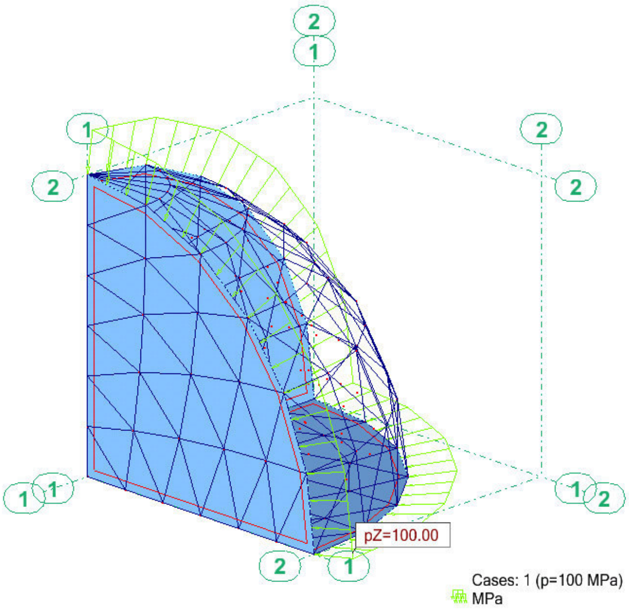
Excellent agreement of results.
This test has been carried out with values of the Poisson's coefficient ranging from 0.3 to 0.499. The relation between -(wa/R)/(ua/L) is always equal to Poisson's coefficient.

VERIFICATION EXAMPLE

Uniform compression of a solid sphere - SSLV02/89

Name of the test : SSLV 02/89
Reference : AFNOR
Specification: Solid sphere - Pressure.

GEOMETRY :



DATA FILE: SSLV02.rtd

COMPARISON:

Node	Compared result	Robot	AFNOR	Difference %
1	Displacement UX (m)	-0.2 e-3	-0.2 e-3	0.0
39	Displacement UY (m)	-0.2 e-3	-0.2 e-3	0.0
14	Displacement UZ (m)	-0.2 e-3	-0.2 e-3	0.0
1	Stress σ_{xx} [MPa]	- 100	- 100	0.0
39	Stress σ_{yy} [MPa]	- 100	- 100	0.0
14	Stress σ_{zz} [MPa]	- 100	- 100	0.0
53	Displacement UX (m)	-0.1 e-3	-0.1 e-3	0.0
61	Displacement UY (m)	-0.1 e-3	-0.1 e-3	0.0
82	Displacement UZ (m)	-0.1 e-3	-0.1 e-3	0.0
53	Stress σ_{xx} [MPa]	- 100	- 100	0.0
61	Stress σ_{yy} [MPa]	- 100	- 100	0.0
82	Stress σ_{zz} [MPa]	- 100	- 100	0.0

CONCLUSION:

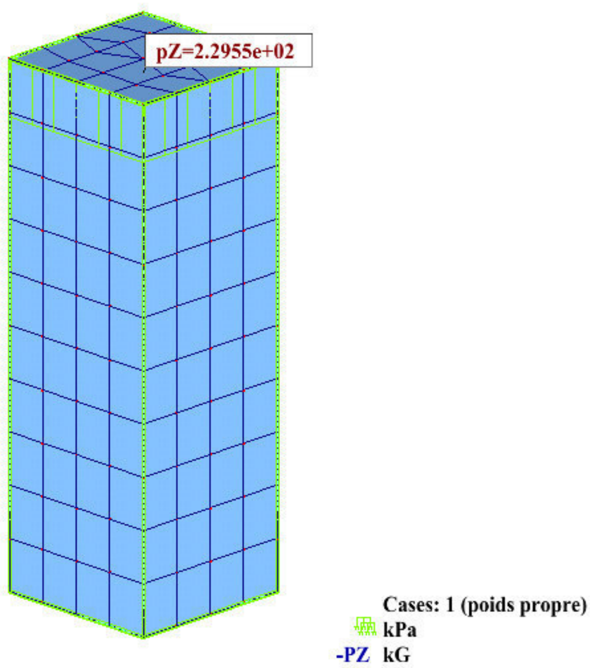
Exact agreement of results.

VERIFICATION EXAMPLE

Tension of a rectangular prism due to self weight - SSLV07/89

Name of the test : SSLV 07/89
Reference : AFNOR
Specification: Solid bar - Tension/compression - Poisson's coefficient.

GEOMETRY :



DATA FILE: SSLV07.rtd

COMPARISON:

Node	Compared result	Robot	AFNOR	Difference %
21	Displacement UZ (m)	-1.72e-6	-1.72e-6	0.0
21, 9	Displacement UZ (m) Δw_{21-9}	0.013e-6	0.014e-6	4.285
271,259	Displacement UX (m) $\Delta u_{271-259}$	0.17e-6	0.17 e-6	0.0
271	Stress σ_{zz} [MPa]	0.2191	0.2290	4.323
146	Stress σ_{zz} [MPa]	0.1147	0.1145	0.0

CONCLUSION:

Results correct.

DYNAMIC ANALYSIS

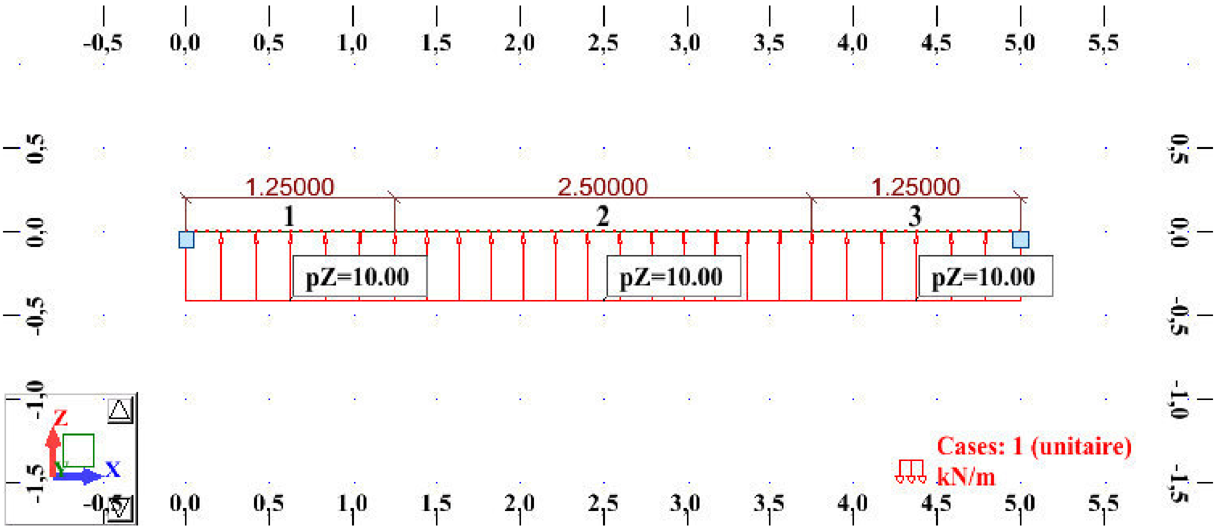
1. BAR STRUCTURES

VERIFICATION EXAMPLE

Slender beam fixed at both ends with different inertia - SDLL03/89

Name of the test: SDLL 03/89
Reference: AFNOR
Specification: Slender beam - Eigen modes - Bending in the plane.

GEOMETRY:



DATA FILE: SDLL03.rtd

COMPARISON:

Node	Compared result	Robot	AFNOR	Difference %
	First bending mode frequency	62.883	63.009	0.200
56 X=0.4375	Eigenvector	1.432 e-2	1.435 e-2	0.209
50 X=0.8125	Eigenvector	3.994 e-2	4.002 e-2	0.200
2 X=1.25	Eigenvector	6.886 e-2	6.899 e-2	0.188
43 X=1.6875	Eigenvector	8.904 e-2	8.922 e-2	0.202
37 X=2.0625	Eigenvector	1.006 e-1	1.008 e-1	0.198
30 X=2.5	Eigenvector	1.056 e-1	1.057 e-1	0.095
31 t=0.0595 s X=2.4375	Vertical displacement UZ (m)	2.231 e-3	2.469 e-3	9.640

CONCLUSION:

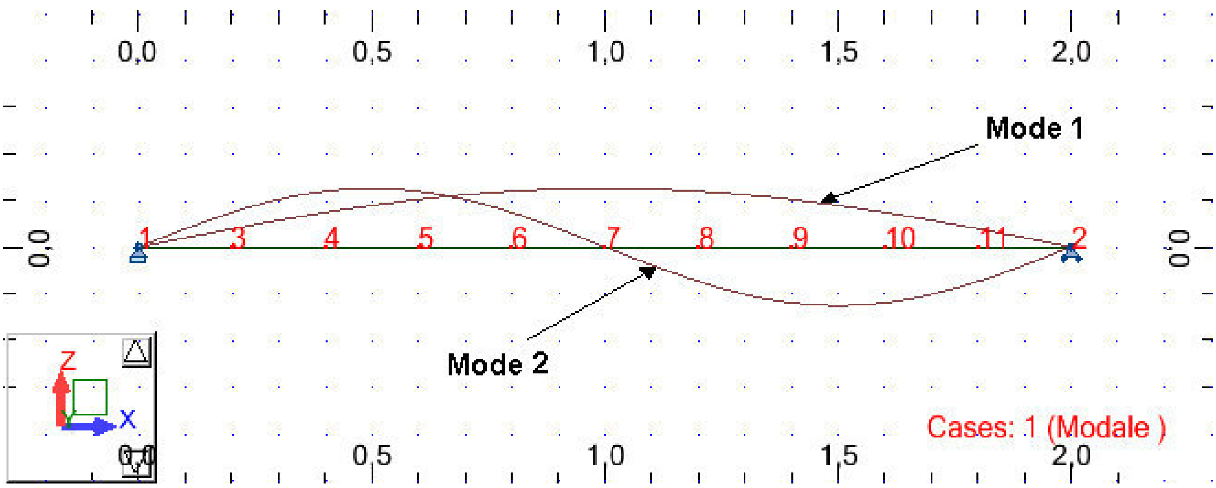
Results correct.

VERIFICATION EXAMPLE

Slender beam supported at both ends subjected to axial load - SDLL05/89

Name of the test: SDLL 05/89
Reference: AFNOR
Specification: Slender beam - Bending in the plane - Eigen modess -Initial stress.

GEOMETRY:



DATA FILE: SDLL05.rtd

COMPARISON:

Case	Frequency	Robot	AFNOR	Difference %
Fx=0	Bending 1	28.694	28.702	0.027
Fx=0	Bending 2	114.701	114.807	0.093
Fx=1 e+5 N	Bending 1	22.428	22.434	0.026
Fx=1 e+5 N	Bending 2	108.981	109.080	0.091

CONCLUSION:

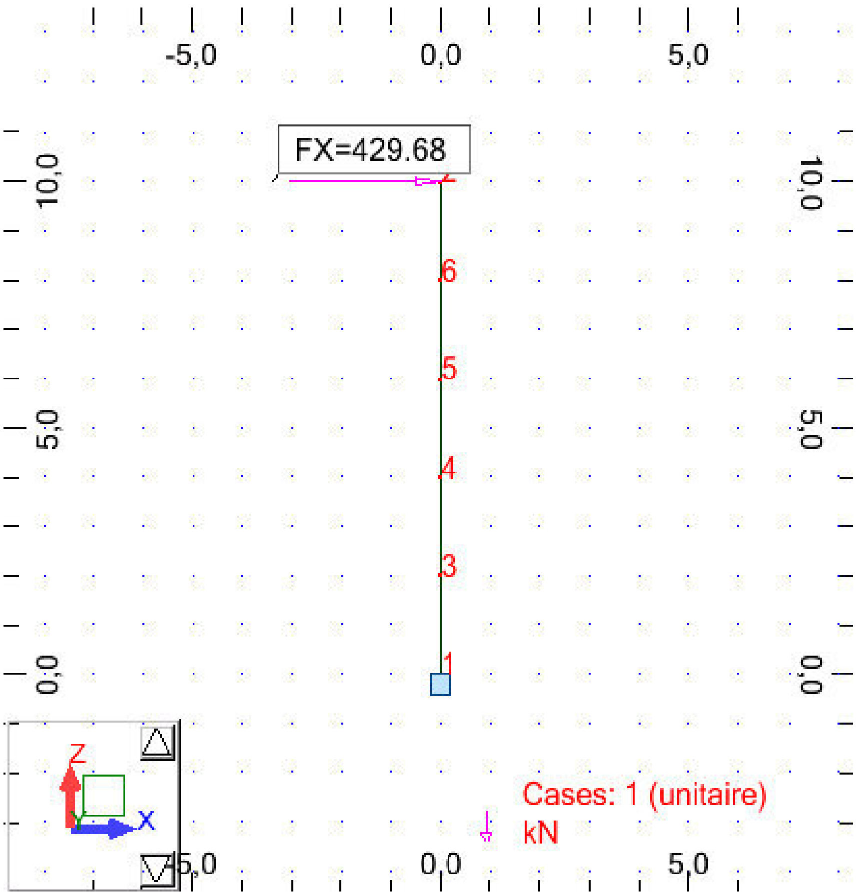
Excellent agreement of results.

VERIFICATION EXAMPLE

Transient analysis of a cantilever under acceleration or imposed load - SDLL 06/89

Name of the test: SDLL 06/89
Reference: AFNOR
Specification: Slender beam - Time history analysis - Plane bending - Imposed force - Imposed acceleration - Modal damping.

GEOMETRY :

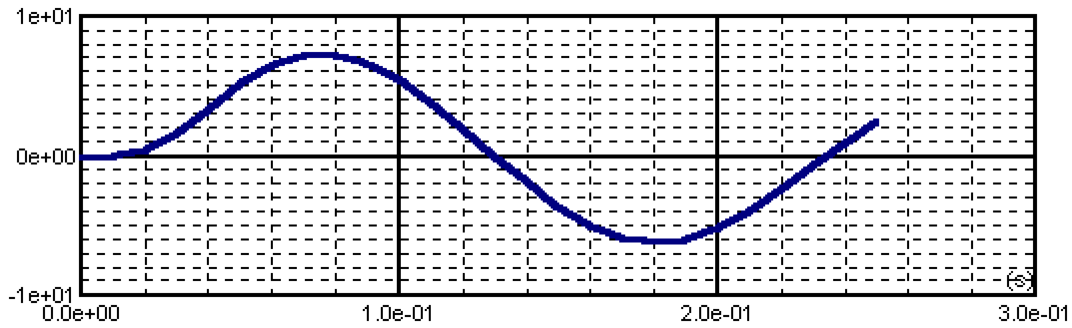


DATA FILE: SDLL 06.rtd

COMPARISON:

Modal analysis		Robot	AFNOR	Difference %
Mode 1	Frequency (Hz)	4.774	4.774	0.0

Time (history) analysis	U_2 - Node 2displacement ux (mm)



Instant (s)	No damping			5% damping		
	Robot	AFNOR	Difference %	Robot	AFNOR	Difference %
0.01	0.065	0.064	*1.563*	0.065	0.065	*0.0*
0.02	0.506	0.504	*0.397*	0.514	0.511	*0.587*
0.03	1.642	1.641	*0.061*	1.679	1.677	*0.119*
0.04	3.351	3.383	*0.946*	3.457	3.489	*0.917*
0.05	5.097	5.157	*1.164*	5.316	5.391	*1.391*
0.06	6.406	6.027	*6.288*	6.765	6.874	*1.586*
0.07	7.113	6.695	*6.244*	7.610	7.774	*2.110*
0.08	7.173	6.749	*6.282*	7.774	7.915	*1.781*
0.09	6.600	6.212	*6.246*	7.245	7.387	*1.922*
0.1	5.463	5.142	*6.243*	6.068	6.198	*2.100*
0.12	2.001	1.880	*6.436*	2.242	2.317	*3.237*
0.14	-1.938	-1.839	*5.383*	-2.367	-2.386	*0.796*
0.16	-4.991	-4.988	*0.060*	-6.150	-6.294	*2.288*
0.18	-6.174	-5.816	*6.155*	-7.784	-7.928	*1.816*
0.2	-5.195	-4.876	*6.542*	-6.699	-6.834	*1.975*

CONCLUSION:

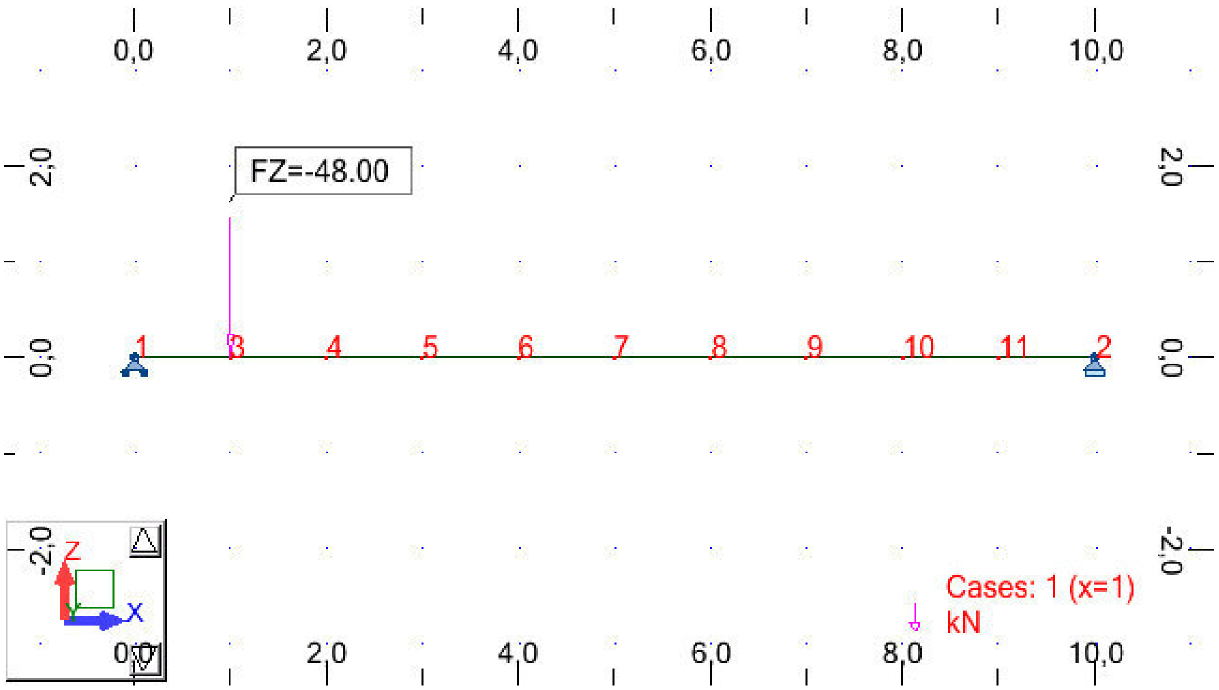
Results correct.

VERIFICATION EXAMPLE

Slender beam supported at both ends subjected to moving load with constant speed-SDLL 07/89

Name of the test: SDLL 07/89
Reference: AFNOR
Specification: Slender beam - Bending in the plane - Eigen modess - Static initial stress.

GEOMETRY:



DATA FILE: SDLL07.rtd

COMPARISON:

Instant	Compared result (node 7)	Robot	AFNOR	Difference %
T=0.1 s	Deflection (m)	-0.04705	-0.04763	1.217
T=0.2 s	Deflection (m)	-0.3206	-0.3235	0.890
T=0.5 s	Deflection (m)	-1.4254	-1.4371	0.814

CONCLUSION:

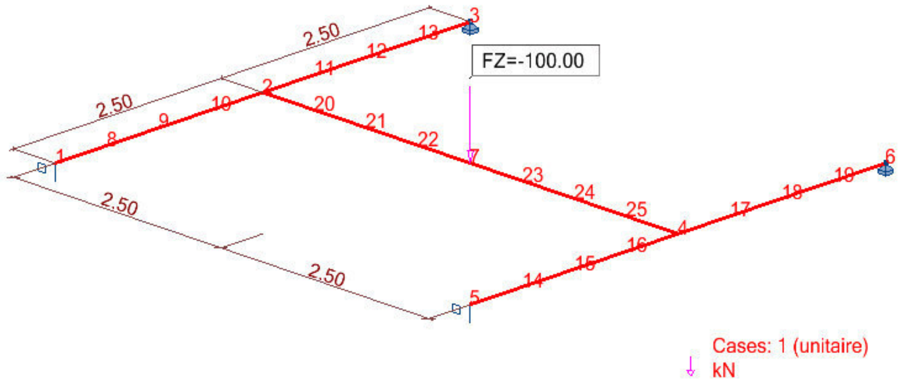
Very good agreement of results.

VERIFICATION EXAMPLE

Plane grillage of beams - SDLL08/89

Name of the test: SDLL 08/89
Reference: AFNOR
Specification: Eigen modes - Transverse bending - Imposed force.

GEOMETRY :



DATA FILE: SDLL 08.rtd

COMPARISON:

Mode analysis		Robot	AFNOR	Difference %
Mode 1	Frequency (Hz)	16.410	16.456	0.280
	Eigenvector w2/(w7-w2)	1.212	1.213	0.0
Mode 3	Frequency (Hz)	37.941	38.196	0.668
	Eigenvector w2/(w7-w2)	-0.412	-0.412	0.0

Harmonic analysis		Robot	AFNOR	Difference %
Node 2	Displacement UZ (m)	-100.54 e-3	- 9.80 e-2	2.592
Node 7	Displacement UZ (m)	-227.74 e-3	- 2.27 e-1	0.0

Time history analysis		Robot	AFNOR	Difference %
Node 2	Displacement UZ (m) comp.966/1001	-98.90 e-3	- 9.80 e-2	0.918
Node 7	Displacement UZ (m) comp.966/1001	-223.76 e-3	- 2.27 e-1	1.427

CONCLUSION:

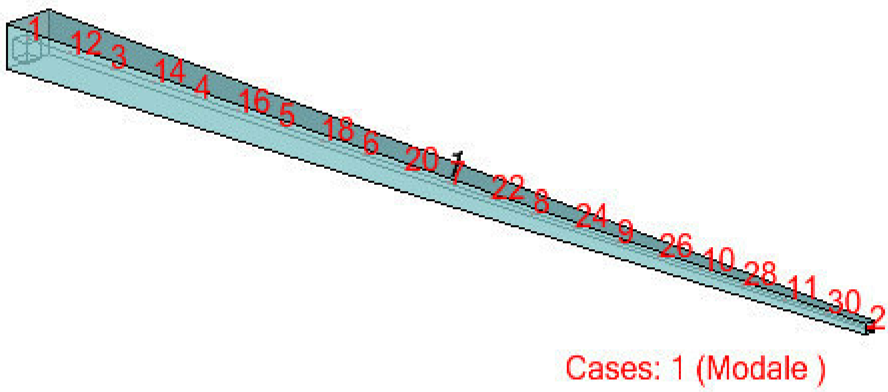
Very good agreement of results.

VERIFICATION EXAMPLE

Slender cantilever fixed at both ends with variable rectangular section - SDLL09/89

Name of the test: SDLL 09/89
Reference: AFNOR
Codification: Eigen modes - Slender beam - Tapered section.

GEOMETRY:



DATA FILE: SDLL09.rtd

COMPARISON:

Type section	Frequency (Hz)	Robot	AFNOR	Difference %
Beta = 4	Mode 1	54.19	54.18	0.02
	Mode 2	171.69	171.94	0.15
	Mode 3	383.05	384.40	0.35
	Mode 4	692.02	697.24	0.75
	Mode 5	1099.65	1112.28	1.14
Beta = 5	Mode 1	56.56	56.55	0.02
	Mode 2	175.57	175.79	0.13
	Mode 3	387.74	389.01	0.33
	Mode 4	697.21	702.36	0.73
	Mode 5	1105.51	1117.63	1.08

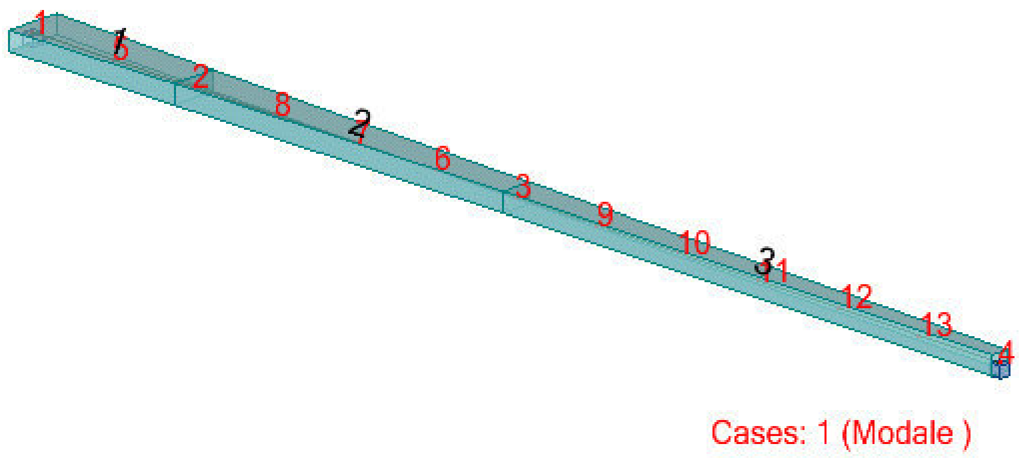
CONCLUSION:
Very good agreement of results.

VERIFICATION EXAMPLE

Slender beam fixed at both ends with variable rectangular section - SDLL10/89

Name of the test: SDLL 10/89
Reference: AFNOR
Specification: Eigen modes - Slender beam - Tapered section - Bending in the plane.

GEOMETRY:



DATA FILE: SDLL10.rtd

COMPARISON:

Frequency (Hz)	Robot	AFNOR	Difference %
Mode 1	145.355	143.303	1.432
Mode 2	398.951	396.821	0.537
Mode 3	780.806	779.425	0.177
Mode 4	1288.503	1289.577	0.083

CONCLUSION:

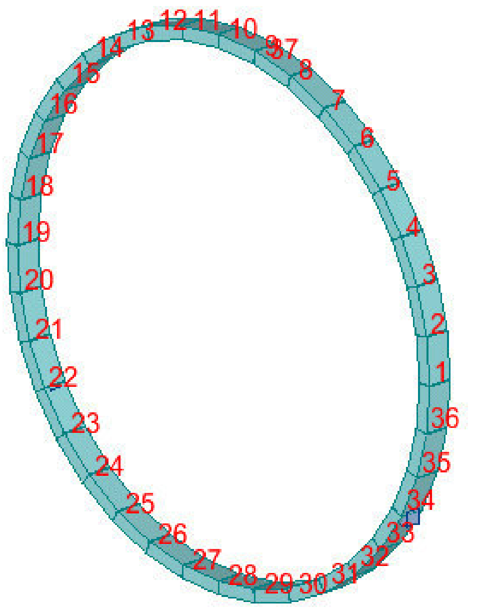
Very good agreement of results.

VERIFICATION EXAMPLE

Ring fixed at two points - SDLL12/89

Name of the test: SDLL 12/89
Reference: AFNOR
Specification: Slender ring - Eigen modes - Bending in the plane.

GEOMETRY:



DATA FILE: SDLL12.rtd

COMPARISON:

Frequency (Hz)	Robot	AFNOR	Difference %
Mode 1	235.888	235.300	0.250
Mode 2	577.053	575.300	0.305
Mode 3	1109.262	1105.700	0.322
Mode 4	1410.008	1405.600	0.314
Mode 5	1755.511	1751.100	0.252
Mode 6	2558.509	2557.000	0.059
Mode 7	2765.514	2801.500	1.285

CONCLUSION:

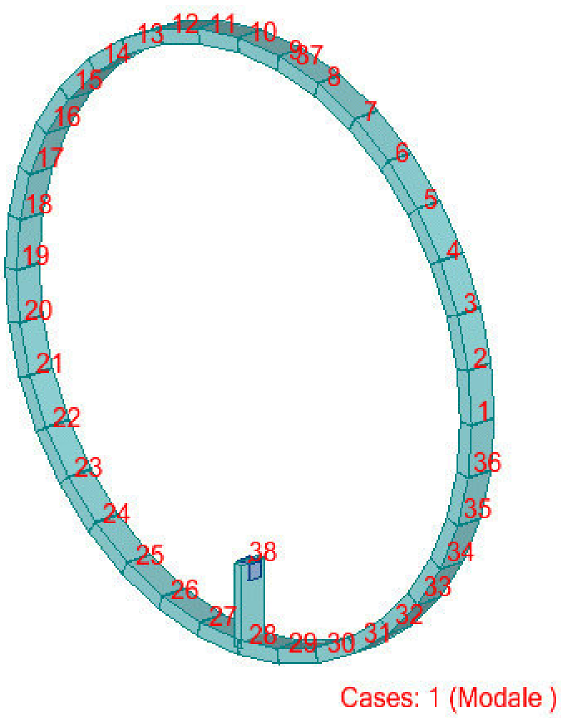
Very good agreement of results.

VERIFICATION EXAMPLE

Ring with flexible support at external point - SDLL13/89

Name of the test: SDLL 13/89
Reference: AFNOR
Specification: Slender ring - Eigen modes - Bending in the plane.

GEOMETRY:



DATA FILE: SDLL13.rtd

COMPARISON:

Frequency (Hz)	Robot	AFNOR	Difference %
Mode 1	28.814	28.800	0.049
Mode 2	189.799	189.300	0.264
Mode 3	269.497	268.800	0.259
Mode 4	640.999	641.000	0.0
Mode 5	684.410	682.000	0.353
Mode 6	1065.192	1063.000	0.206

CONCLUSION:

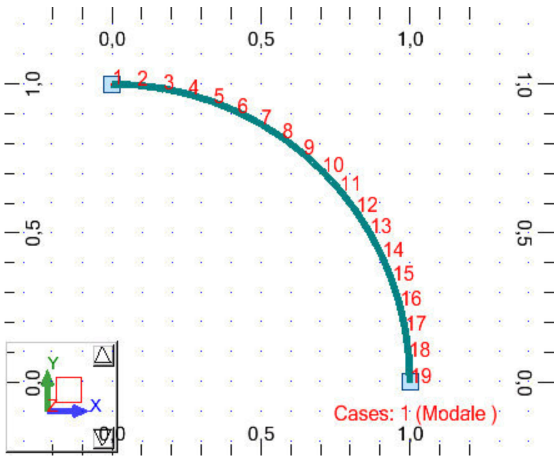
Excellent agreement of results.

VERIFICATION EXAMPLE

Eigenmode of a thin-walled tube section - SDLL14/89

Name of the test: SDLL 14/89
Reference: AFNOR
Specification: Eigen modes - Slender curved beam - Bending in the plane - Transversal bending.

GEOMETRY:



DATA FILE: SDLL14a.rtd; SDLL14b.rtd ; SDLL14c.rtd

COMPARISON:

Case		Frequency (Hz)	Robot	AFNOR	Difference %
L=0	a	Mode 1	44.178	44.230	0.118
		Mode 2	119.675	119.000	0.567
		Mode 3	126.058	125.000	0.846
		Mode 4	226.490	227.000	0.225
L=0.6	b	Mode 1	33.240	33.400	0.479
		Mode 2	94.227	94.000	0.241
		Mode 3	98.955	100.000	1.045
		Mode 4	183.372	180.000	1.873
L=2	c	Mode 1	17.660	17.900	1.341
		Mode 2	24.432	24.800	1.484
		Mode 3	24.949	25.300	1.387
		Mode 4	26.723	27.000	1.026

CONCLUSION:

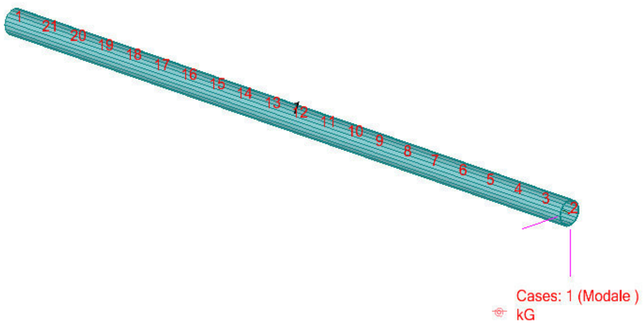
Very good agreement of results.

VERIFICATION EXAMPLE

Slender cantilever with mass eccentricity at the end of it - SDLL15/89

Name of the test: SDLL 15/89
Reference: AFNOR
Specification: Eigen modes - Slender beam - Bending and torsion - Bending in the plane - Transversal bending - Mass at the end of the cantilever.

GEOMETRY:



DATA FILE: SDLL15a.rtd; SDLL15b.rtd;

COMPARISON:

Case		Frequency (Hz)	Robot	AFNOR	Difference %
Yc=0	a	Flexion 1, 2	1.655	1.650	0.303
		Flexion 3, 4	16.055	16.070	0.093
		Flexion 5, 6	49.866	50.020	0.308
		Traction 1	76.473	76.470	0.0
		Torsion 1	80.469	80.470	0.0
		Flexion 9, 10	102.512	103.20	0.667

Case		Frequency (Hz)	Robot	AFNOR	Difference %
Yc=1	b	Flexion x,z + torsion 1	1.636	1.636	0.0
		Flexion x,y + traction 2	1.642	1.642	0.0
		Flexion x,y + traction 3	13.446	13.460	0.104
		Flexion x,z + torsion 4	13.587	13.590	0.022
		Flexion x,z + torsion 5	28.847	28.900	0.183
		Flexion x,y + traction 6	31.929	31.960	0.097
		Flexion x,z + torsion 7	61.291	61.610	0.518
		Flexion x,y + traction 8	63.737	63.930	0.302

CONCLUSION:

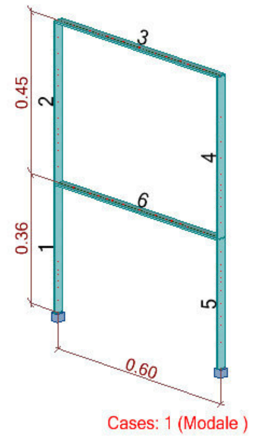
Excellent agreement of results.

VERIFICATION EXAMPLE

Symmetrical frame bending - SDLX01/89

Name of the test: SDLX 01/89
Reference: AFNOR
Specification: Slender beam - Bending in the plane - Eigen modes.

GEOMETRY:



DATA FILE: SDLX01.rtd

COMPARISON:

Frequency (Hz)	Robot	AFNOR	Difference %
Mode 1	8.75	8.80	0.57
Mode 2	29.35	29.40	0.17
Mode 3	43.71	43.80	0.21
Mode 4	56.12	56.30	0.32
Mode 5	95.87	96.20	0.34
Mode 6	102.37	102.60	0.22
Mode 7	146.63	147.10	0.32
Mode 8	174.38	174.80	0.24
Mode 9	178.34	178.80	0.26
Mode 10	205.56	206.00	0.21
Mode 11	265.80	266.40	0.23
Mode 12	319.35	320.00	0.20
Mode 13	334.45	335.00	0.16

CONCLUSION:

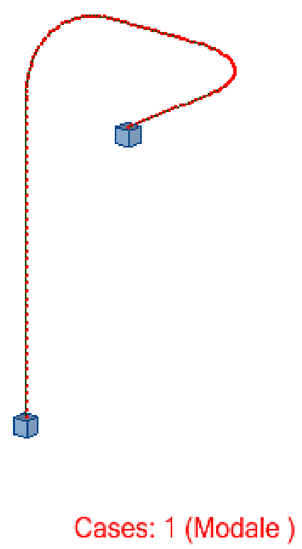
Excellent agreement of results.

VERIFICATION EXAMPLE

Hovgaard's problem - stress in the 3D pipe - bending - SDLX02/89

Name of the test: SDLX 02/89
Reference: AFNOR
Specification: Eigen modes - Bending in the plane -Transversal bending - Slender curved beam.

GEOMETRY:



DATA FILE: SDLX02.rtd

COMPARISON:

Frequency (Hz)	Robot	AFNOR	Difference %
Mode 1	10.25	10.18	0.69
Mode 2	19.96	19.54	2.15
Mode 3	25.08	25.47	1.53
Mode 4	47.71	48.09	0.79
Mode 5	52.35	52.86	0.96
Mode 6	84.26	75.94	10.96
Mode 7	86.51	80.11	7.99
Mode 8	126.57	122.34	3.46
Mode 9	130.86	123.15	6.26

CONCLUSION:
5 first modes give correct results.

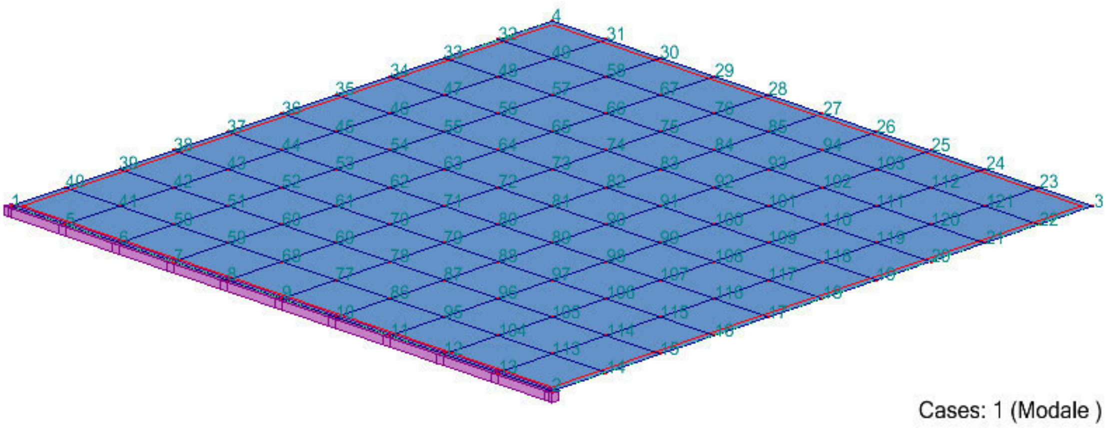
2. PLATES/SHELLS STRUCTURES

VERIFICATION EXAMPLE

Cantilever plate - SDLS01/89

Name of the test: SDLS 01/89
Reference: AFNOR
Specification: Square thin plate with one edge fixed

GEOMETRY:



DATA FILE: SDLS01.rtd

COMPARISON:

Frequency (Hz)	Robot	AFNOR	Difference %
Mode 1	8.6655	8.7266	0.700
Mode 2	21.2450	21.3042	0.278
Mode 3	53.6890	53.5542	0.252
Mode 4	68.5652	68.2984	0.391
Mode 5	77.9989	77.7448	0.327
Mode 6	137.1204	136.0471	0.789

CONCLUSION:

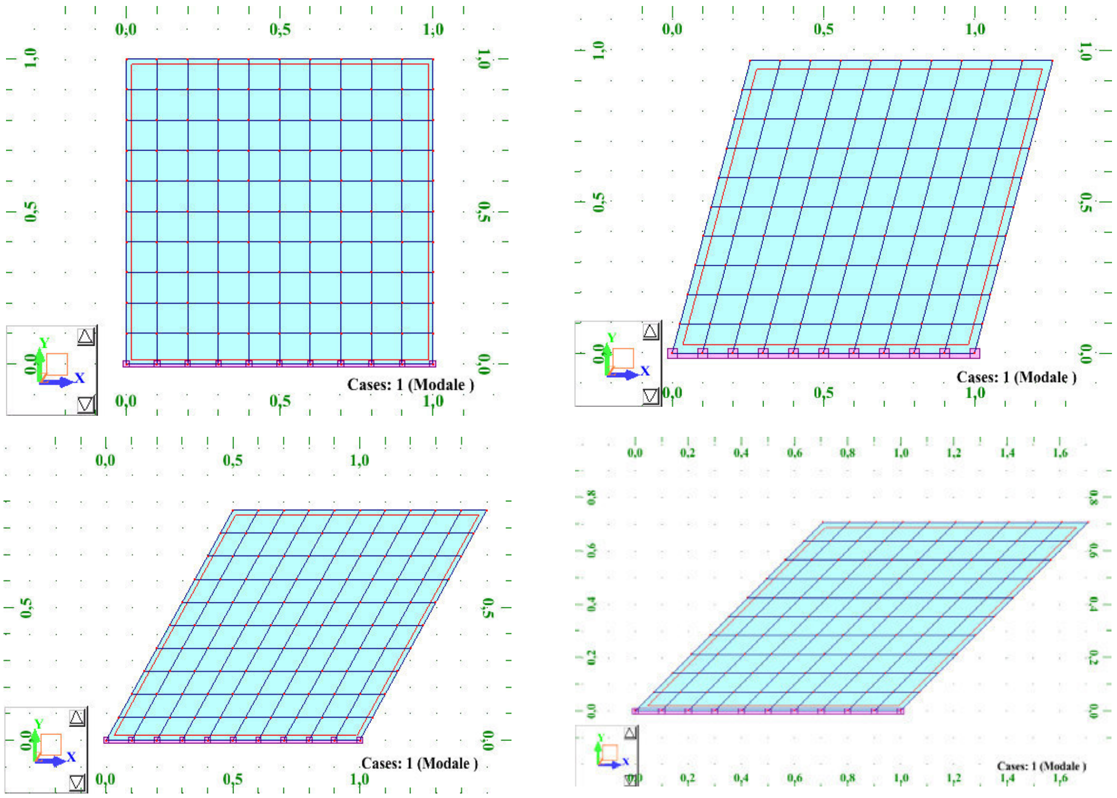
Excellent agreement of results.

VERIFICATION EXAMPLE

Lozenge - shaped thin plate with one edge fixed - SDLS02/89

Name of the test: SDLS 02/89
Reference: AFNOR
Specification: Lozenge - shaped thin plate with one edge fixed

GEOMETRY:



DATA FILE: SDLS02a.rtd; SDLS02b.rtd; SDLS02c.rtd; SDLS02d.rtd

COMPARISON:

ALPHA [deg]	Case	Frequency (Hz)	Robot	AFNOR	Difference %
0	a	Mode 1	8.6655	8.7266	0.700
		Mode 2	21.2450	21.3042	0.278
15	b	Mode 1	8.9422	8.9990	0.631
		Mode 2	21.7167	22.1714	2.051
30	c	Mode 1	9.7945	9.8987	1.053
		Mode 2	23.4749	25.4651	7.815
45	d	Mode 1	11.16	11.15	0.001
		Mode 2	27.58	27.00	2.148

CONCLUSION:

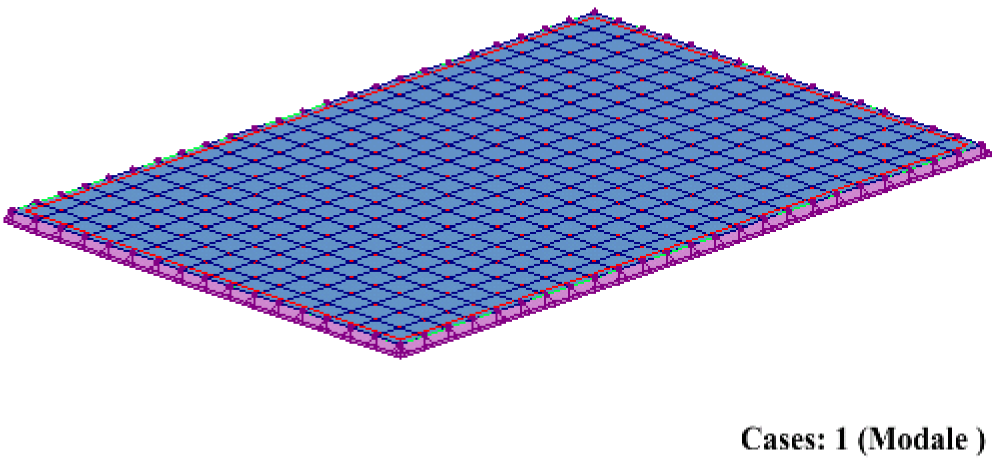
According to "Guide de ..." accuracy of AFNOR is 3%, but for higher mode numbers it can be less accurate.

VERIFICATION EXAMPLE

Simply supported rectangular thin plate - SDLS03/89

Name of the test: SDLS 03/89
Reference: AFNOR
Specification: Simply supported rectangular thin plate

GEOMETRY:



DATA FILE: SDLS03.rtd

COMPARISON:

Frequency (Hz)	Robot	AFNOR	Difference %
Mode 1	35.72	35.63	0.25
Mode 2	68.84	68.51	0.47
Mode 3	110.85	109.62	1.12
Mode 4	124.63	123.32	1.06
Mode 5	143.99	142.51	1.04
Mode 6	199.84	197.32	1.27

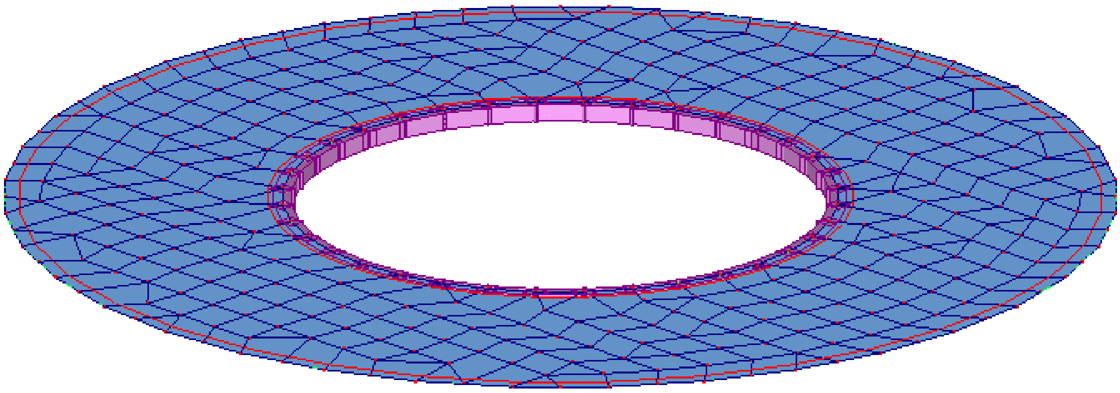
CONCLUSION:
Very good agreement of results.

VERIFICATION EXAMPLE

Circular plate with fixed inner edge - SDLS04/89

Name of the test: SDLS 04/89
Reference: AFNOR
Specification: Circular plate fixed at inner edge

GEOMETRY:



DATA FILE: SDLS04.rtd

COMPARISON:

Mode	Frequency fij (Hz)		Robot	AFNOR	Difference %
1	i=0	j=0	79.48	79.26	0.28
2 and 3	i=1	j=0	80.98	81.09	0.14
4 and 5	i=2	j=0	89.57	89.63	0.06
6 and 7	i=3	j=0	113.23	112.79	0.39
18	i=0	j=1	526.08	518.85	1.39
19 and 20	i=1	j=1	533.34	528.61	0.90
21 and 22	i=2	j=1	567.26	559.09	1.46
23 and 24	i=3	j=1	621.91	609.7	2.00

i = number of nodal diameters
j = number of nodal circles

CONCLUSION:

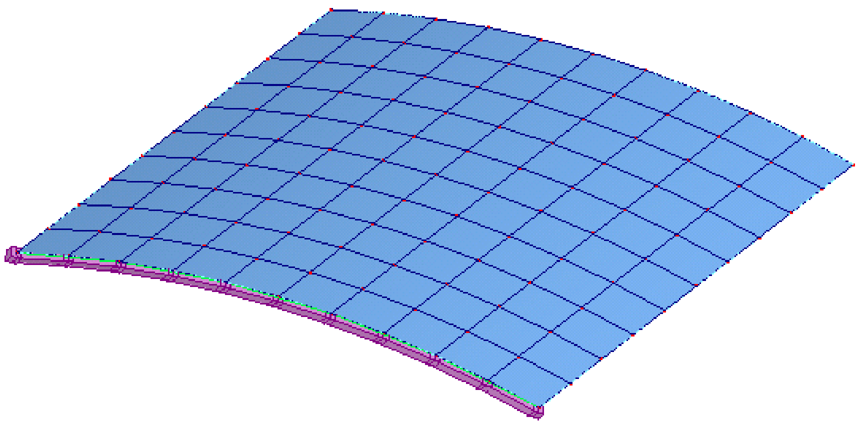
Very good agreement of results.

VERIFICATION EXAMPLE

Compressor blade: thin shell - SDLS05/89

Name of the test: SDLS 05/89
Reference: AFNOR
Specification: Thin shell with one edge fixed

GEOMETRY:



DATA FILE: SDLS05.rtd

COMPARISON:

Frequency (Hz)	Robot	AFNOR	Difference %
Mode 1	86.12	85.60	0.61
Mode 2	138.47	134.50	2.95
Mode 3	250.00	259.00	3.47
Mode 4	346.52	351.00	1.28
Mode 5	389.68	395.00	1.35
Mode 6	547.34	531.00	3.08

CONCLUSION:

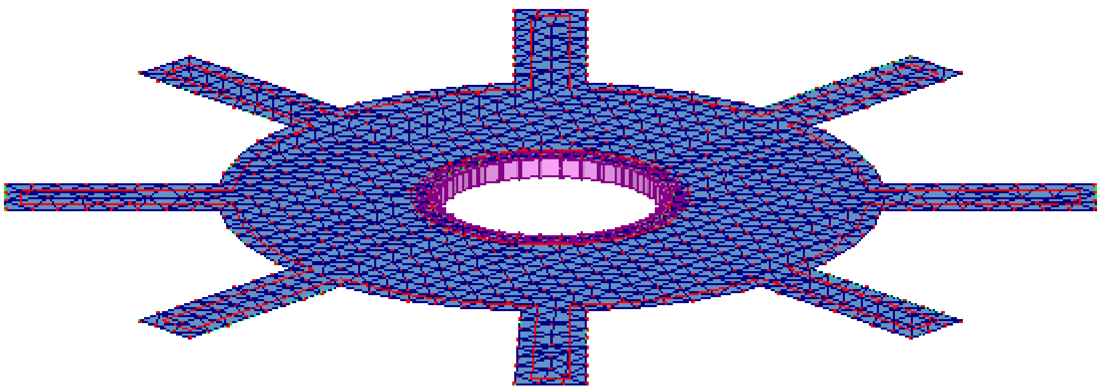
According to "Guide de..." accuracy of AFNOR is 3%, but for the higher modes it is less precise.

VERIFICATION EXAMPLE

Modal analysis of plate - SDLS06/89

Name of the test: SDLS 06/89
Reference: AFNOR
Specification: Eigen modes - Thin plate

GEOMETRY:



Cases: 1 (Modale)

DATA FILE: SDLS06.rtd

COMPARISON:

Mode	Frequency [Hz]		Difference %
	Robot	AFNOR	
Torsion Flexion	286.91	295.10	2.85
	370.74	361.10	2.60
	399.97	390.50	2.43
	969.81	971.00	0.12
	1671.14	1663.00	0.49
	2178.83	2189.00	0.46
	2598.50	2627.00	1.08

CONCLUSION:

Good agreement of results.

THERMOMECHANICAL ANALYSIS

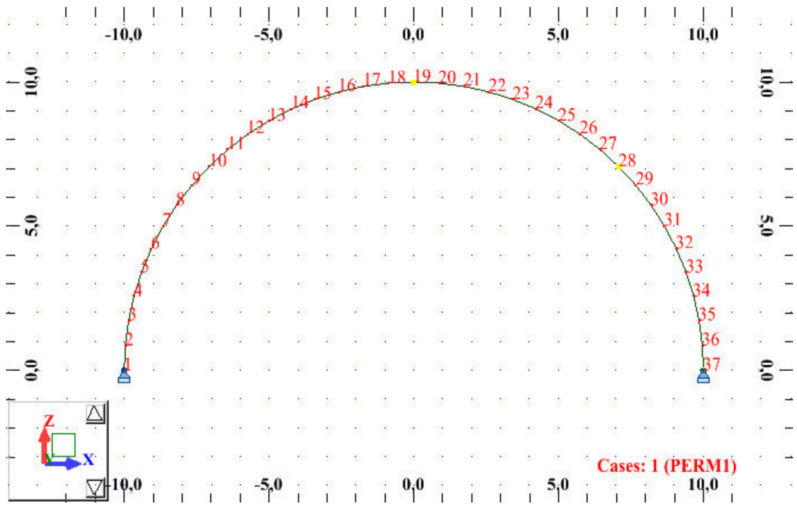
1. BAR STRUCTURES

VERIFICATION EXAMPLE

Arch with 2 pinned supports - HSLL01/89

Name of the test: HSLL 01/89
Reference: AFNOR
Specification: Thin-walled arch - Temperature gradient - Pinned supports.

GEOMETRY:



DATA FILE: HSLL01.rtd

COMPARISON:

Position	Value type	Robot	AFNOR	Difference %
Alpha=90 Node 37	Bending moment(Nm)	0	0	0.0
	Normal force (N)	-209.397	0	
	Shear force (N)	-4790.303	-4792.000	0.035
Alpha=45 Node 28	Bending moment (Nm)	33905.060	33883.000	0.065
	Normal Force (N)	-3239.121	-3388.000	4.394
	Shear force (N)	-3535.385	-3388.000	4.350
Alpha=0 Node 19	Bending moment (Nm)	47948.778	47918.000	0.064
	Normal Force(N)	-4790.303	-4792.000	0.035
	Shear force (N)	-209.397	0	

CONCLUSIONS:

Results correct.
The results have been obtained from the average of forces from 2 bars met in a node.
Normal force (shear force) for alpha=90° (alpha=0°) is not equal to 0 because the arch consists of linear segments.
Nevertheless the value is still reliable.

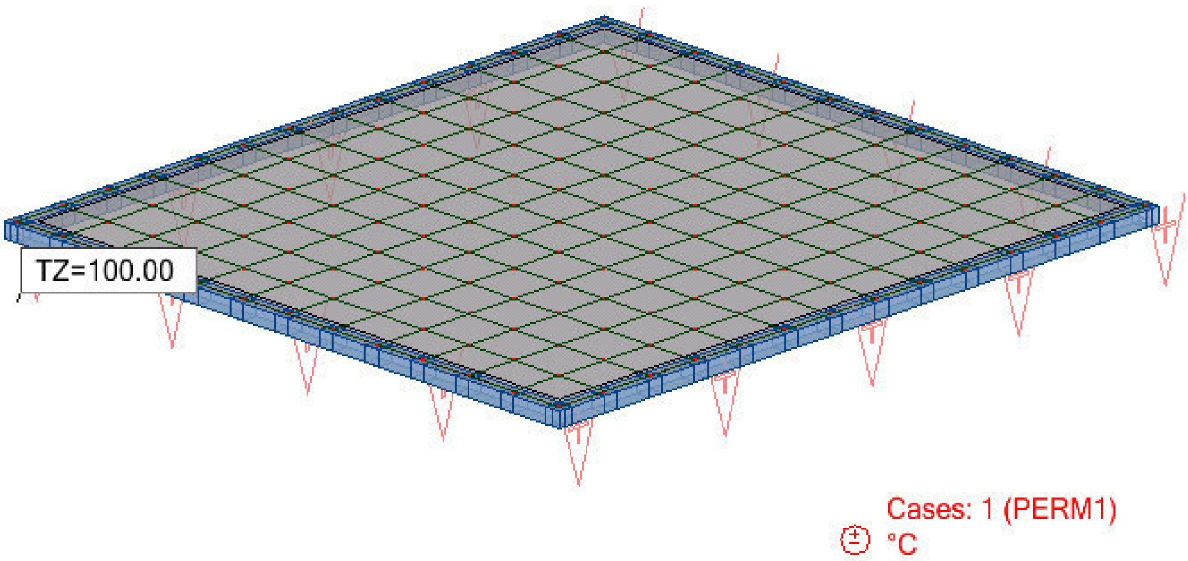
2. PLATES/SHELLS STRUCTURES

VERIFICATION EXAMPLE

Thin plate deformed according to spherical curve - HSLS01/89

Name of the test: HSLS 01/89
Reference: AFNOR
Specification: Thin plate - Thermal gradient - Fixed support.

GEOMETRY:



DATA FILE HSLS01.rtd

COMPARISON:

Position	Type of the value	Robot	AFNOR	Difference %
On the edges	Bending moment (Nm/m)	2380.95	2380.95	0.0
On the edges	Maximum stress (Pa)	142.857 e+6	142.185 e+6	0.47

CONCLUSION:

Excellent agreement of results.

CONCLUSIONS

The results and accuracy achieved in verification examples confirm the quality and reliability of Robot. This state-of-the-art structural analysis and design software gives excellent accuracy within the applied solution method.

Autodesk®
Robot™ Structural Analysis Professional

VERIFICATION MANUAL

FOR EU CODES

March 2014

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)

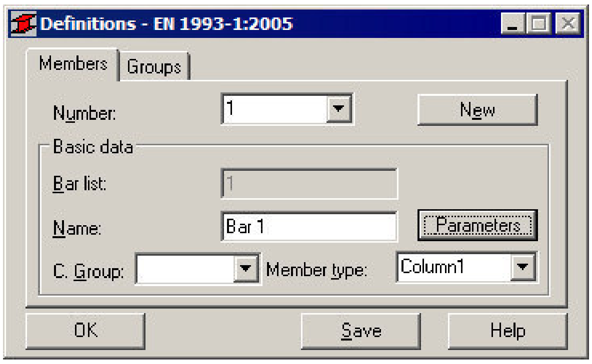
VERIFICATION EXAMPLE 1
- Axial compression

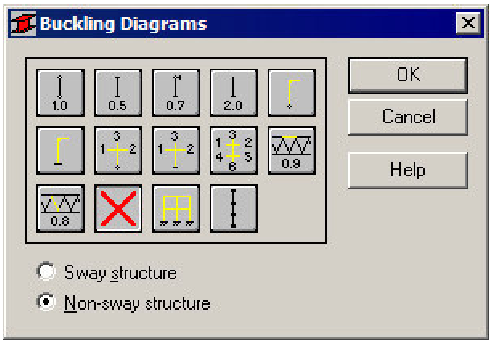
Example taken from Designer's Guide to EN 1993-1-1
L.Gardner and D.A.Nethercot, Thomas Telford Publishing, 2005

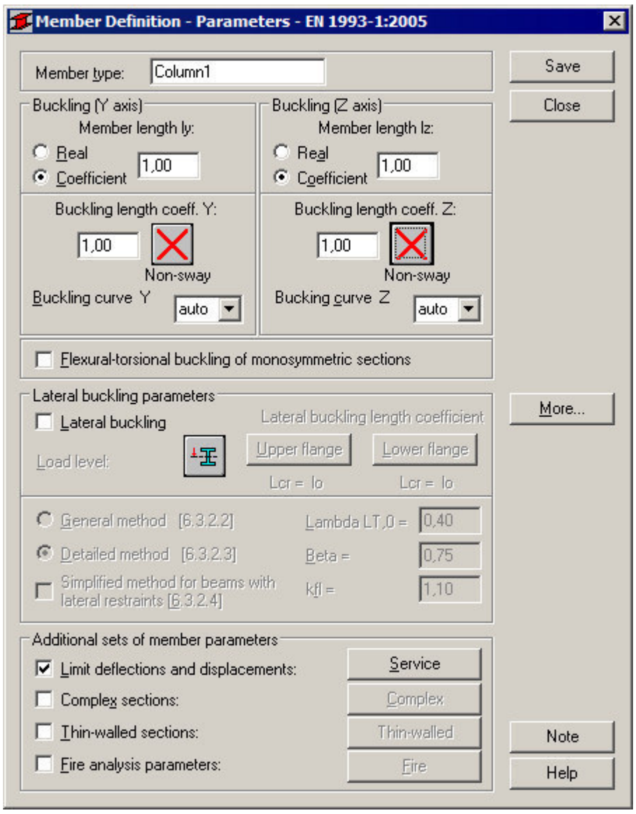
TITLE:
Axial compression (Example 6.2 page 44).

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.

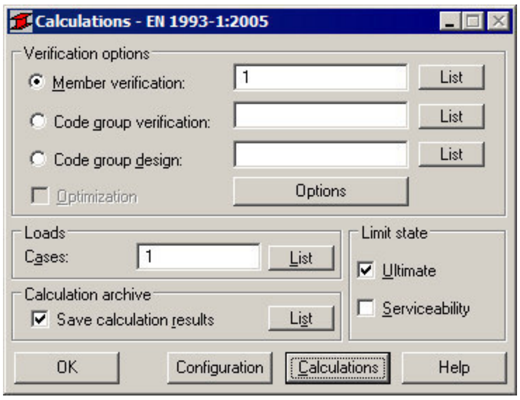
SOLUTION:
Define a new type of member. For analysed member pre-defined type of member COLUMN may be initially opened. Press the *Parameters* button in DEFINITIONS/MEMBERS tab, which opens MEMBER DEFINITION – PARAMETERS dialog. Type a new name **Column 1** in the *Member Type* editable field. Then, press *Buckling Length coefficient Y* icon and select the twelvth icon (*no bucklling*). For *Z* direction press *Buckling Length coefficient Z* and choose the same icon. Save the newly-created type of member.



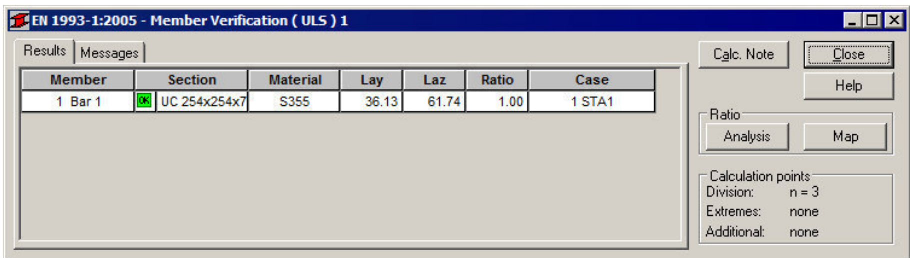




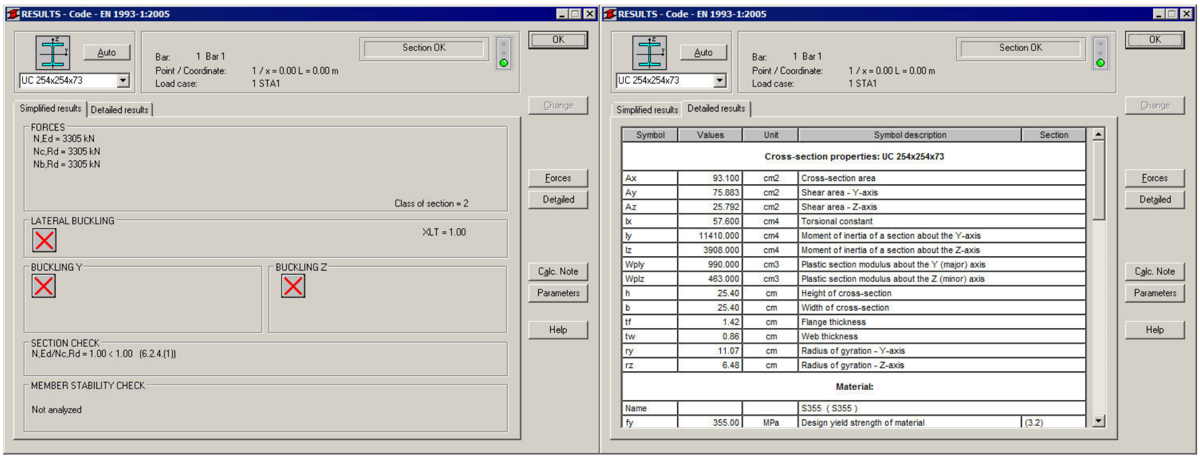
In the CALCULATIONS dialog set *Member Verification* option for member 1 and switch off *Limit State – Serviceability* (only Ultimate Limit state will be analysed). Now, start the calculations by pressing *Calculations* button.



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.



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.



STEEL DESIGN

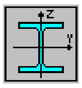
CODE: *EN 1993-1:2005, Eurocode 3: Design of steel structures.*
ANALYSIS TYPE: Member Verification

CODE GROUP:
MEMBER: 1 Bar 1 **POINT:** 1 **COORDINATE:** x = 0.00 L = 0.00 m

---

LOADS:
Governing Load Case: 1 STA1

MATERIAL:
S355 (S355) fy = 355.00 MPa



SECTION PARAMETERS: UC 254x254x73

h=25.40 cm	gM0=1.00	gM1=1.00	
b=25.40 cm	Ay=75.883 cm2	Az=25.792 cm2	Ax=93.100 cm2
tw=0.86 cm	Iy=11410.000 cm4	Iz=3908.000 cm4	Ix=57.600 cm4
tf=1.42 cm	Wply=990.000 cm3	Wplz=463.000 cm3	

INTERNAL FORCES AND CAPACITIES:
N,Ed = 3305 kN
Nc,Rd = 3305 kN
Nb,Rd = 3305 kN

Class of section = 2

LATERAL BUCKLING PARAMETERS:

BUCKLING PARAMETERS:

About Y axis: About Z axis:

VERIFICATION FORMULAS:
Section strength check:
N,Ed/Nc,Rd = 1.00 < 1.00 (6.2.4.(1))

Section OK !!!

COMPARISON:

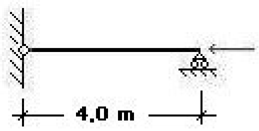
Resistance, interaction expression	Robot	Handbook
1. design compression resistance of the cross+section $N_{c,Rd}$	3305	3305

VERIFICATION EXAMPLE 2
- Axial compression with buckling

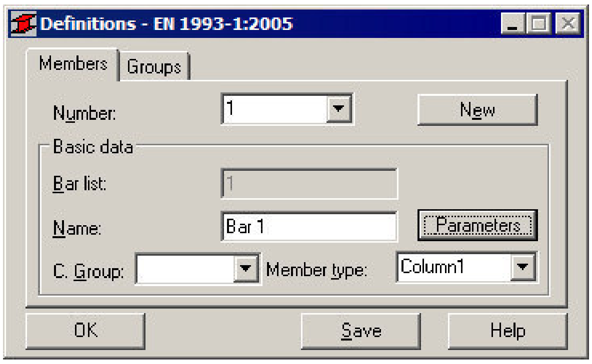
Example taken from Designer's Guide to EN 1993-1-1
L.Gardner and D.A.Nethercot, Thomas Telford Publishing, 2005

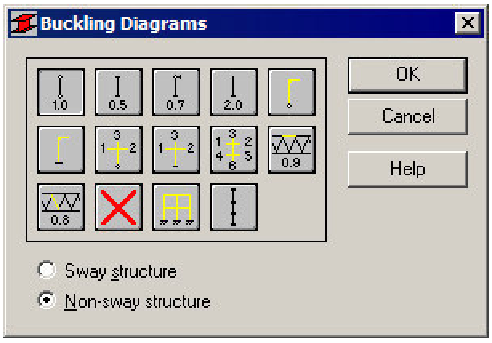
TITLE:
Buckling resistance of a compression member (Example 6.7 page 66).

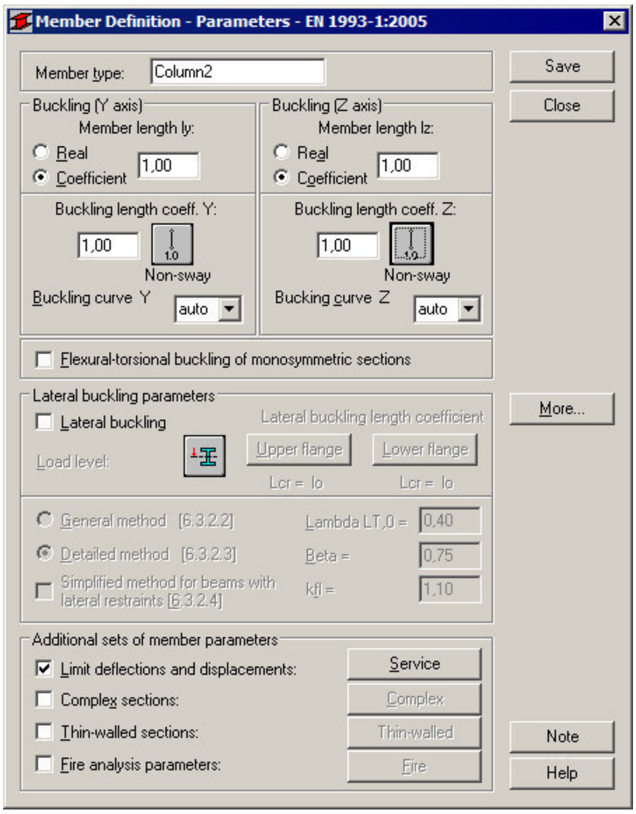
SPECIFICATION:
The member shown below has pinned boundary conditions. The design compression force N = 1630 kN is checked for the assumed circular hollow section CHS 244,5x10, steel grade S275.



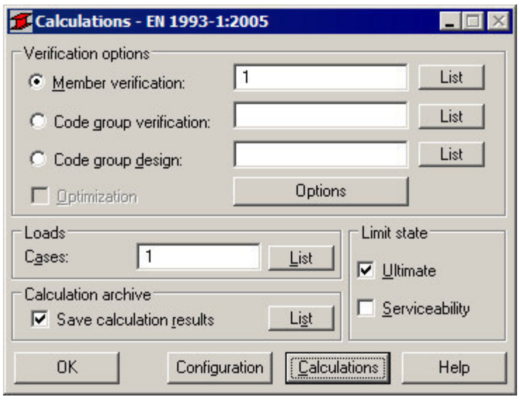
SOLUTION:
Define a new type of member. For analysed member pre-defined type of member COLUMN may be initially opened. Press the *Parameters* button in DEFINITIONS/MEMBERS tab, which opens MEMBER DEFINITION – PARAMETERS dialog. Type a new name **Column 2** in the *Member Type* editable field. The *Buckling Length coefficient Y* and *Z* are set to the buckling length *1.0*. Save the newly-created type of member.



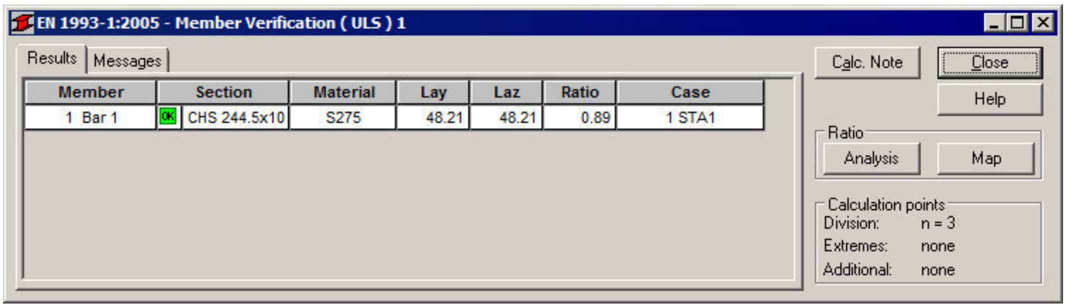




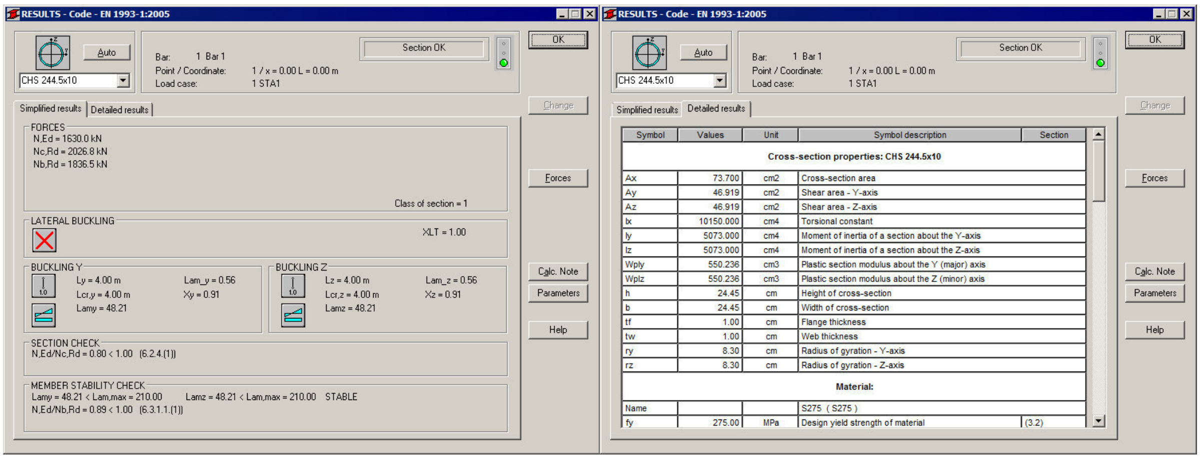
In the CALCULATIONS dialog set *Member Verification* option for member 1 and switch off *Limit State – Serviceability* (only Ultimate Limit state will be analysed). Now, start the calculations by pressing *Calculations* button.



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.



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.



STEEL DESIGN

CODE: *EN 1993-1:2005, Eurocode 3: Design of steel structures.*
ANALYSIS TYPE: Member Verification

CODE GROUP:
MEMBER: 1 Bar 1 **POINT:** 1 **COORDINATE:** x = 0.00 L = 0.00 m

---

LOADS:
Governing Load Case: 1 STA1

MATERIAL:
S275 (S275) fy = 275.00 MPa

SECTION PARAMETERS: CHS 244.5x10

h=24.45 cm	gM0=1.00	gM1=1.00	
b=24.45 cm	Ay=46.919 cm2	Az=46.919 cm2	Ax=73.700 cm2
tw=1.00 cm	Iy=5073.000 cm4	Iz=5073.000 cm4	Ix=10150.000 cm4
tf=1.00 cm	Wply=550.236 cm3	Wplz=550.236 cm3	

INTERNAL FORCES AND CAPACITIES:
N,Ed = 1630.0 kN
Nc,Rd = 2026.8 kN
Nb,Rd = 1836.5 kN

Class of section = 1

LATERAL BUCKLING PARAMETERS:

BUCKLING PARAMETERS:

About Y axis:		About Z axis:	
Ly = 4.00 m	Lam_y = 0.56	Lz = 4.00 m	Lam_z = 0.56
Lcr,y = 4.00 m	Xy = 0.91	Lcr,z = 4.00 m	Xz = 0.91
Lamy = 48.21		Lamz = 48.21	

VERIFICATION FORMULAS:
Section strength check:
N,Ed/Nc,Rd = 0.80 < 1.00 (6.2.4.(1))
Global stability check of member:
Lambda,y = 48.21 < Lambda,max = 210.00 Lambda,z = 48.21 < Lambda,max = 210.00 STABLE
N,Ed/Nb,Rd = 0.89 < 1.00 (6.3.1.1.(1))

Section OK !!!

COMPARISON:

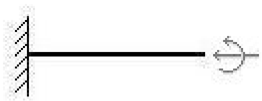
Resistance, interaction expression	Robot	Handbook
1. cross-section compression resistance $N_{c.Rd}$	2026.8	2026.8
2. non-demensional slenderness for flexural buckling Lambda	0,56	0,56

VERIFICATION EXAMPLE 3
- Combined compression and bending

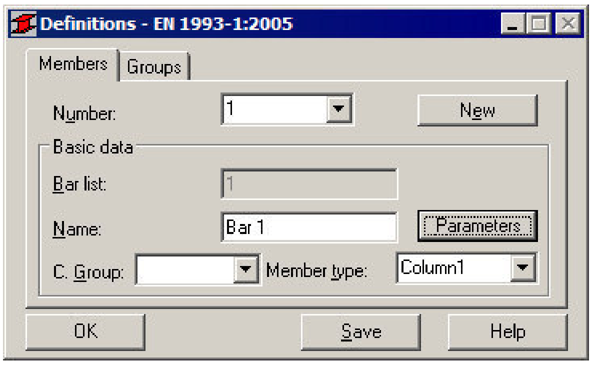
Example taken from Designer's Guide to EN 1993-1-1
L.Gardner and D.A.Nethercot, Thomas Telford Publishing, 2005

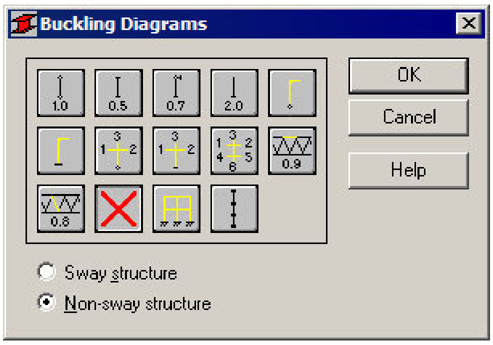
TITLE:
Combined compression and bending (Example 6.6 page 57).

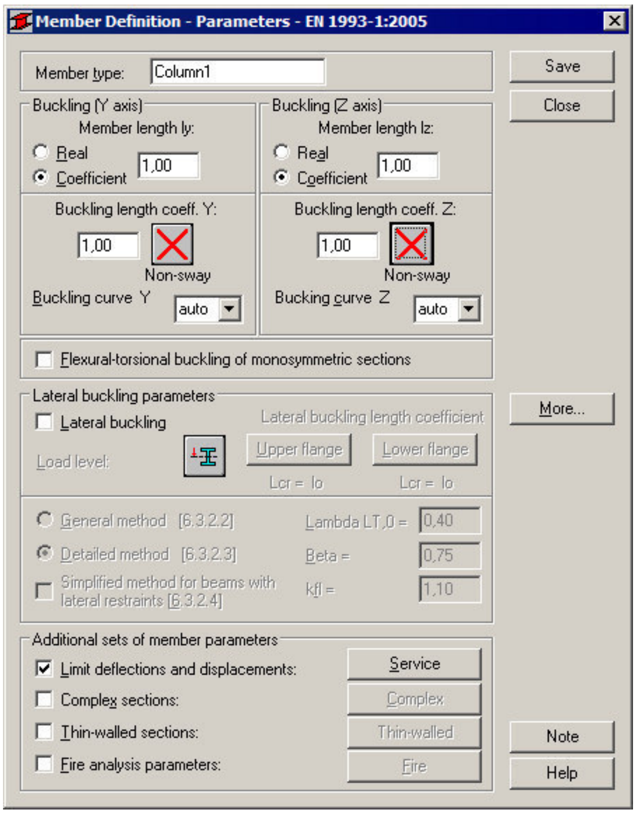
SPECIFICATION:
The member carry combined major axis bending moment and an axial force. The assumed section UB 457x191x98 in grade S235 steel is checked to determine the maximum bending moment in the presence of an axial force N = 1400 kN.



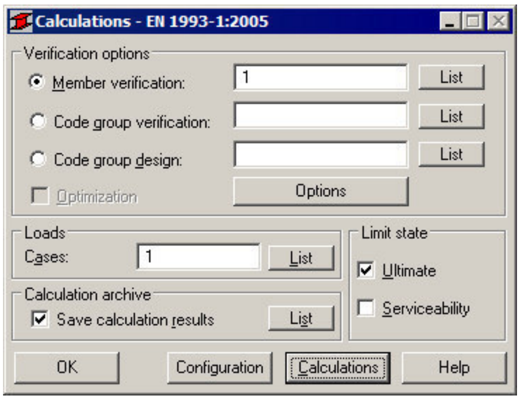
SOLUTION:
Define a new type of member. For analysed member pre-defined type of member COLUMN may be initially opened. Press the *Parameters* button in DEFINITIONS/MEMBERS tab, which opens MEMBER DEFINITION – PARAMETERS dialog. Type a new name **Column 1** in the *Member Type* editable field. Then, press *Buckling Length coefficient Y* icon and select the twelvth icon (*no bucklling*). For *Z* direction press *Buckling Length coefficient Z* and choose the same icon. Save the newly-created type of member.



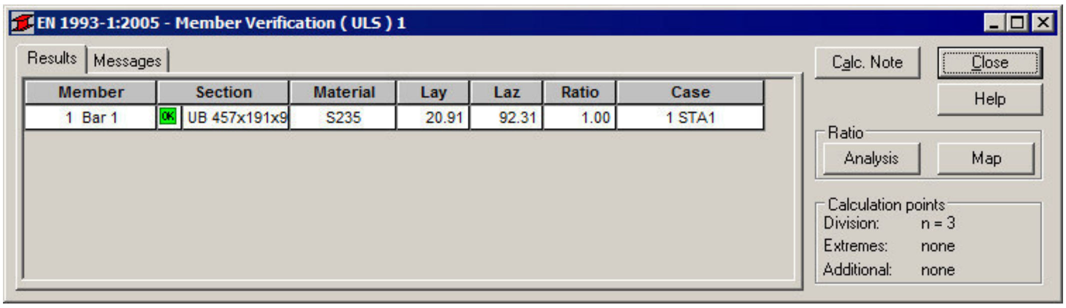




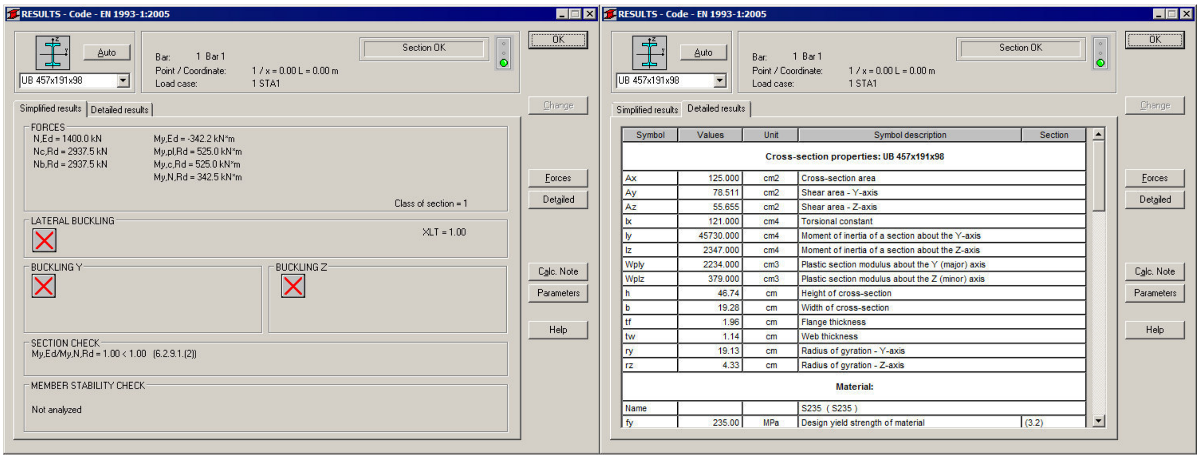
In the CALCULATIONS dialog set *Member Verification* option for member 1 and switch off *Limit State – Serviceability* (only Ultimate Limit state will be analysed). Now, start the calculations by pressing *Calculations* button.



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.



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.



STEEL DESIGN

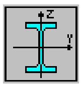
CODE: *EN 1993-1:2005, Eurocode 3: Design of steel structures.*
ANALYSIS TYPE: Member Verification

CODE GROUP:
MEMBER: 1 Bar 1 **POINT:** 1 **COORDINATE:** x = 0.00 L = 0.00 m

---

LOADS:
Governing Load Case: 1 STA1

MATERIAL:
S235 (S235) fy = 235.00 MPa



SECTION PARAMETERS: UB 457x191x98

h=46.74 cm	gM0=1.00	gM1=1.00	
b=19.28 cm	Ay=78.511 cm2	Az=55.655 cm2	Ax=125.000 cm2
tw=1.14 cm	Iy=45730.000 cm4	Iz=2347.000 cm4	Ix=121.000 cm4
tf=1.96 cm	Wply=2234.000 cm3	Wplz=379.000 cm3	

INTERNAL FORCES AND CAPACITIES:

N,Ed = 1400.0 kN	My,Ed = -342.2 kN*m
Nc,Rd = 2937.5 kN	My,pl,Rd = 525.0 kN*m
Nb,Rd = 2937.5 kN	My,c,Rd = 525.0 kN*m
	My,N,Rd = 342.5 kN*m

Class of section = 1

LATERAL BUCKLING PARAMETERS:

BUCKLING PARAMETERS:

About Y axis: About Z axis:

VERIFICATION FORMULAS:
Section strength check:
N,Ed/Nc,Rd = 0.48 < 1.00 (6.2.4.(1))
My,Ed/My,c,Rd = 0.65 < 1.00 (6.2.5.(1))
My,Ed/My,N,Rd = 1.00 < 1.00 (6.2.9.1.(2))

Section OK !!!

COMPARISON:

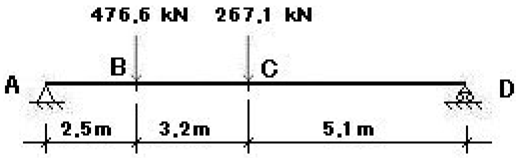
Resistance, interaction expression	Robot	Handbook
1. plastic moment resistance $M_{pl,y,.Rd}$	525,0	524,5
2. reduced plastic moment resistance $M_{N,y,.Rd}$	342,5	342,2

VERIFICATION EXAMPLE 4
- Bending with lateral buckling

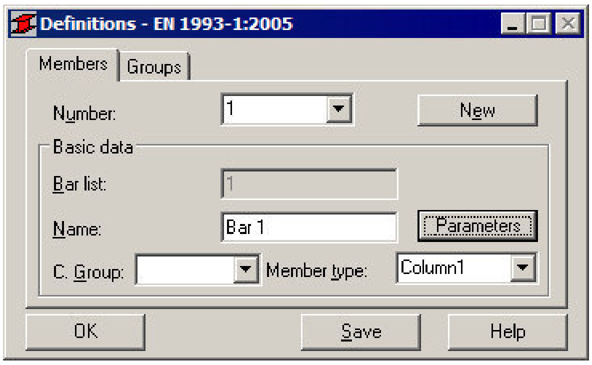
Example taken from Designer's Guide to EN 1993-1-1
L.Gardner and D.A.Nethercot, Thomas Telford Publishing, 2005

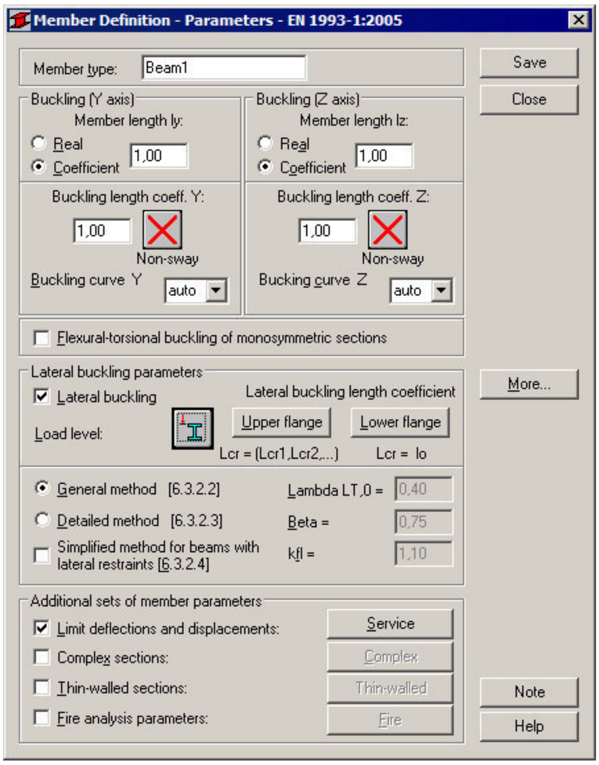
TITLE:
Lateral torsional buckling resistance (Example 6.8 page 74).

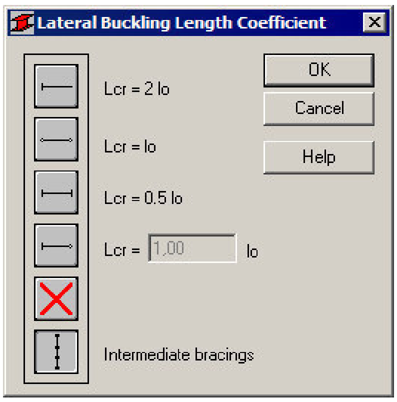
SPECIFICATION:
Simply supported primary beam supports two secondary beams, represented with the concentrated load as shown below. The secondary beams create full lateral restraint of the primary beam web at these points. Section UB 762x267x173 is checked in grade S275 steel. The loads given are at the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



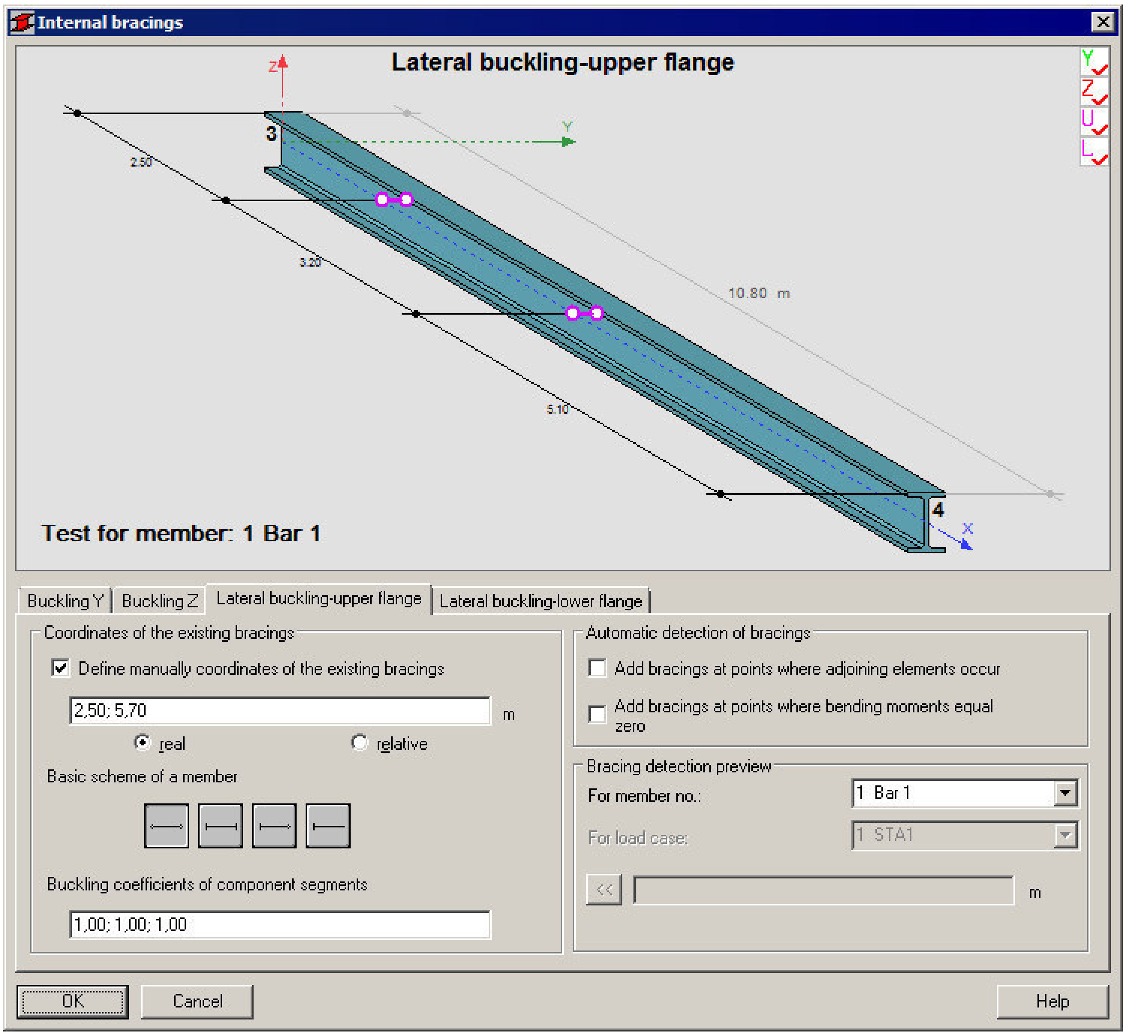
SOLUTION:
Define a new type of member. For analysed member pre-defined type of member BEAM may be initially opened. It can be set in *Member type* combo-box. Press the *Parameters* button in DEFINITION-MEMBERS tab, which opens MEMBER DEFINITION – PARAMETERS dialog. Type a new name **Beam 1** in the *Member Type* editable field.
Select the radio button *General method (6.3.2.2.)* in the *Lateral buckling parameters.*



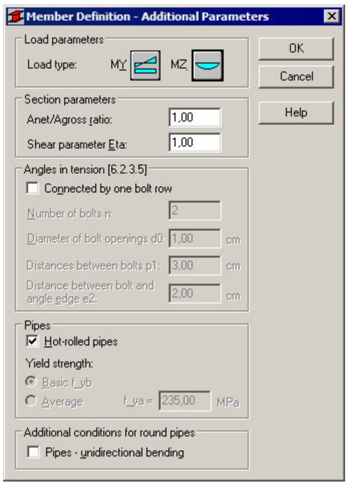


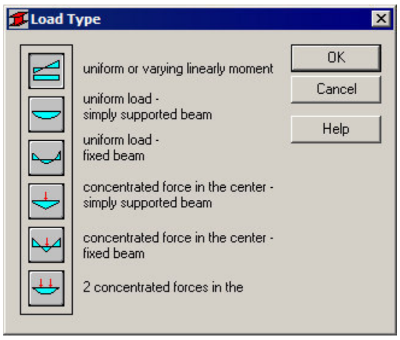


Then, press *Lateral buckling coefficient – Upper flange* icon and select the last icon (*Intermediate bracing*) that opens *Internal bracing* dialog. Define the coordinates of the existing bracing, change to *real* length radio button, type in: *2.50 5,70* (m) in the *Coordinate of the existing bracing* edit box. Close dialog by pressing OK. Do not change lateral buckling length for the lower flange.

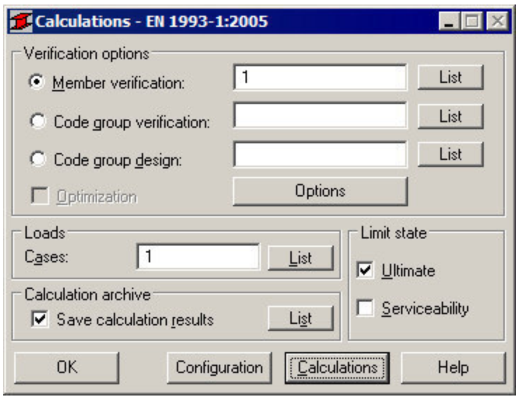


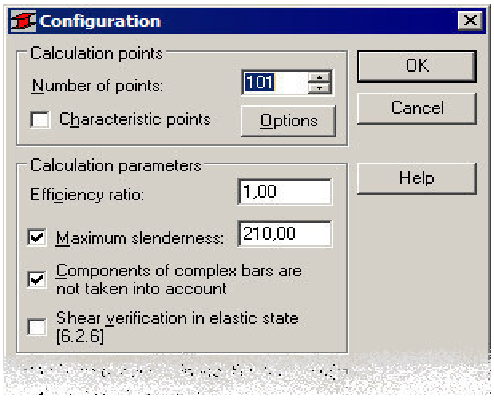
For defining appropriate load type diagram, press *More* button. Choose the icon for Load type Y and double-click the first icon (*Uniform moment and varying linearly*) in *Load Type* dialog.





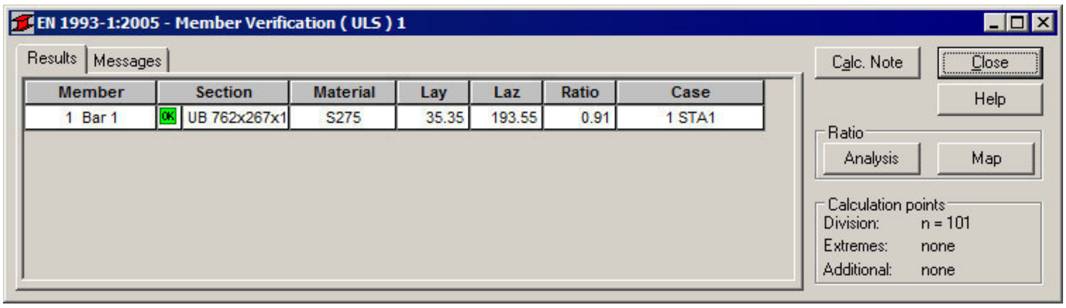
Save the newly-created type of member.
In the CALCULATIONS dialog set *Member Verification* option for member 1 and switch off *Limit State – Serviceability* (only Ultimate Limit state will be analysed). Call configuration dialog and set number of calculation points to 101.



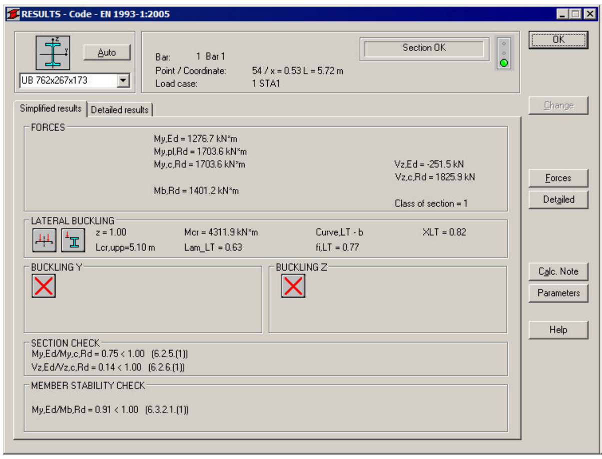


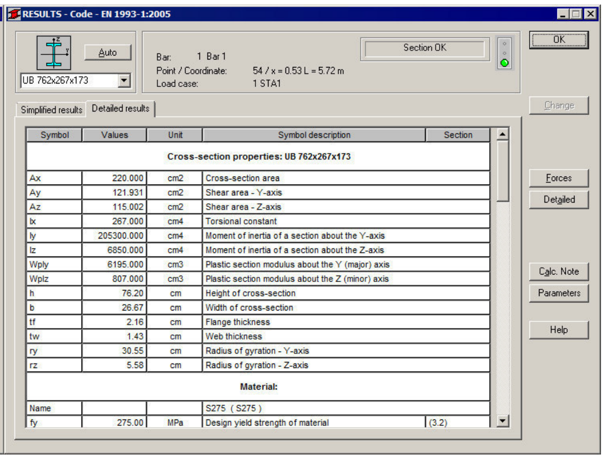
Now, start the calculations by pressing *Calculations* button.

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.



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.





STEEL DESIGN

CODE: *EN 1993-1:2005, Eurocode 3: Design of steel structures.*
ANALYSIS TYPE: Member Verification

CODE GROUP:
MEMBER: 1 Bar 1 **POINT:** 54 **COORDINATE:** x = 0.53 L = 5.72 m

---

LOADS:
Governing Load Case: 1 STA1

MATERIAL:
S275 (S275) fy = 275.00 MPa

SECTION PARAMETERS: UB 762x267x173

h=76.20 cm	gM0=1.00	gM1=1.00	
b=26.67 cm	Ay=121.931 cm2	Az=115.002 cm2	Ax=220.000 cm2
tw=1.43 cm	Iy=205300.000 cm4	Iz=6850.000 cm4	Ix=267.000 cm4
tf=2.16 cm	Wply=6195.000 cm3	Wplz=807.000 cm3	

INTERNAL FORCES AND CAPACITIES:

My,Ed = 1276.7 kN*m
My,pl,Rd = 1703.6 kN*m
My,c,Rd = 1703.6 kN*m
Vz,Ed = -251.5 kN
Vz,c,Rd = 1825.9 kN
Mb,Rd = 1401.2 kN*m
Class of section = 1

LATERAL BUCKLING PARAMETERS:

z = 1.00	Mcr = 4311.9 kN*m	Curve,LT - b	XLT = 0.82
Lcr,upp=5.10 m	Lam_LT = 0.63	fi,LT = 0.77	

BUCKLING PARAMETERS:

About Y axis: About Z axis:

VERIFICATION FORMULAS:

Section strength check:
My,Ed/My,c,Rd = 0.75 < 1.00 (6.2.5.(1))
Vz,Ed/Vz,c,Rd = 0.14 < 1.00 (6.2.6.(1))
Global stability check of member:
My,Ed/Mb,Rd = 0.91 < 1.00 (6.3.2.1.(1))

Section OK !!!

COMPARISON: Critical segment CD

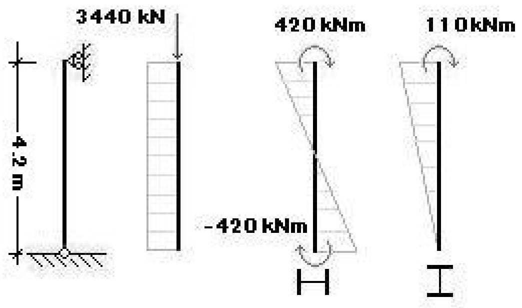
Resistance, interaction expression	Robot	Handbook
1. Critical moment for lateral-torsional buckling Mcr	4311,9	4311
2. Reduction factor for lateral-torsional buckling X_{LT}	0,82	0,82

VERIFICATION EXAMPLE 5
- Combined bi-axial bending and compression

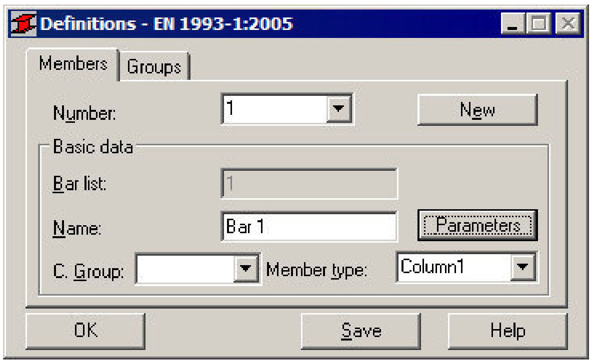
Example taken from Designer's Guide to EN 1993-1-1
L.Gardner and D.A.Nethercot, Thomas Telford Publishing, 2005

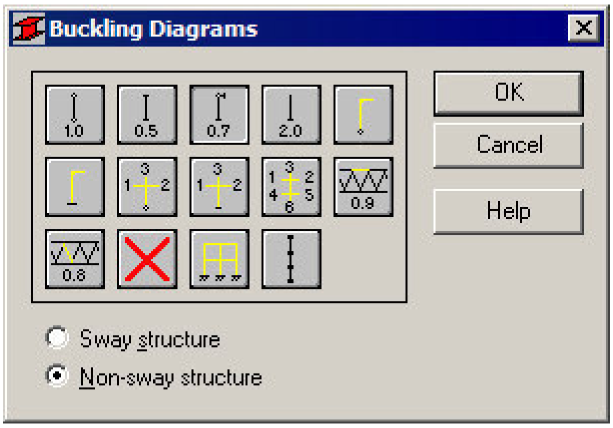
TITLE:
Combined bi-axial bending and compression (Example 6.10 page 89).

SPECIFICATION:
The model represents a column in a multistory building. The column frame is moment resisting in-plna and pinned out-of-plane, with diagonal bracing in both directions. The modeled bar shown below is pin ended about y-y and z-z axes. The bar is subjected to the compressive force and bending in major axis due to horizontal forces, in minor axis due to eccentric axial load. Section H 305x305x240 is checked in grade S275 steel. The loads are given at ultimate limit state.

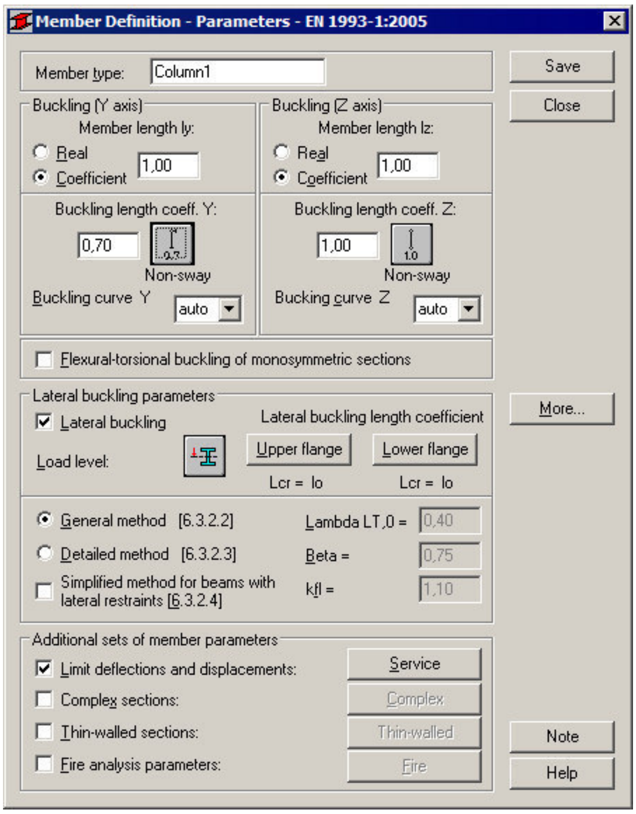


SOLUTION:
Define a new type of member. For analysed member pre-defined type of member COLUMN may be initially opened. Press the *Parameters* button in DEFINITIONS/MEMBERS tab, which opens MEMBER DEFINITION – PARAMETERS dialog. Type a new name **Column 1** in the *Member Type* editable field. Then, press *Buckling Length coefficient Y* icon and select the third icon (*0.7*). For *Z* direction let it defined default *1.0*.



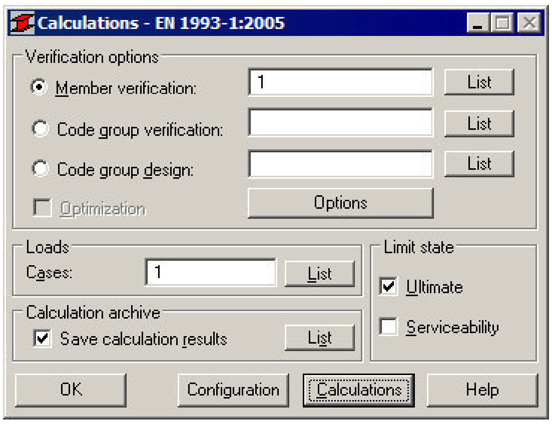


Set *Lateral buckling* checkbox.
Select the radio button *General method (6.3.2.2.)* in the *Lateral buckling parameters.*

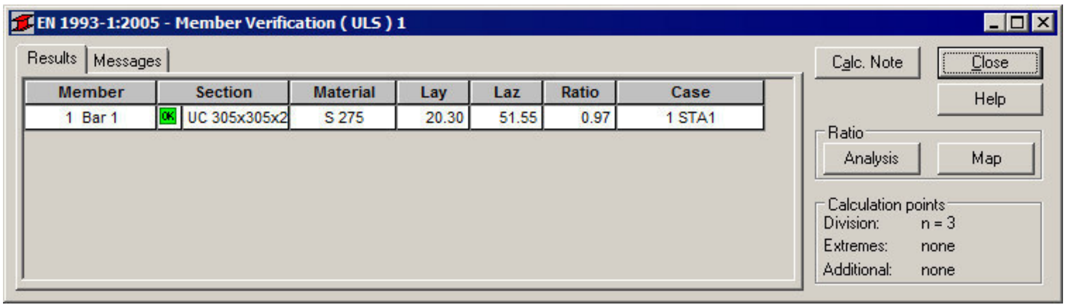


Save the newly-created type of member.

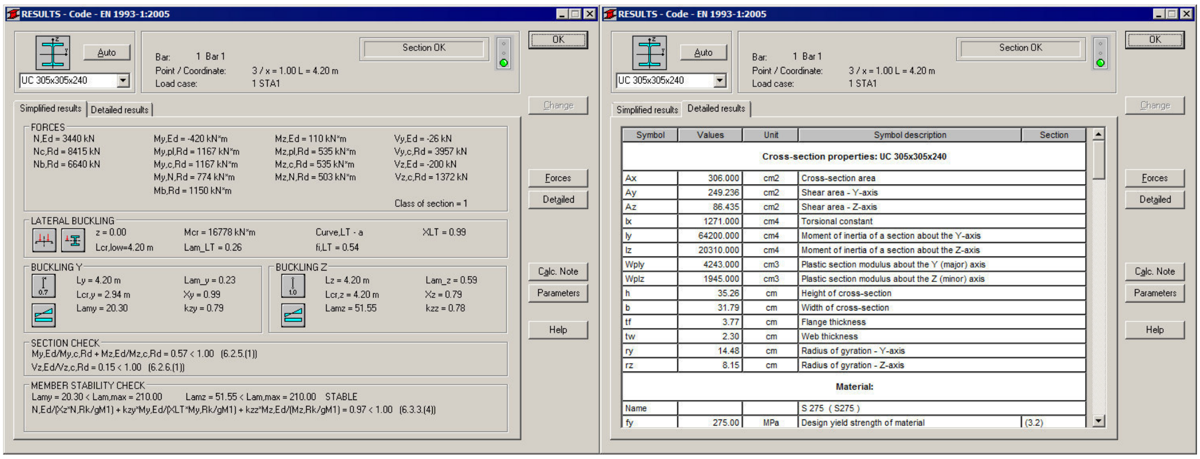
In the CALCULATIONS dialog set *Member Verification* option for member 1 and switch off *Limit State – Serviceability* (only Ultimate Limit state will be analysed). Now, start the calculations by pressing *Calculations* button.



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.



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.



STEEL DESIGN

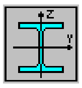
CODE: *EN 1993-1:2005, Eurocode 3: Design of steel structures.*
ANALYSIS TYPE: Member Verification

CODE GROUP:
MEMBER: 1 Bar 1 **POINT:** 3 **COORDINATE:** x = 1.00 L = 4.20 m

---

LOADS:
Governing Load Case: 1 STA1

MATERIAL:
S 275 (S275) fy = 275.00 MPa



SECTION PARAMETERS: UC 305x305x240

h=35.26 cm	gM0=1.00	gM1=1.00	
b=31.79 cm	Ay=249.236 cm2	Az=86.435 cm2	Ax=306.000 cm2
tw=2.30 cm	Iy=64200.000 cm4	Iz=20310.000 cm4	Ix=1271.000 cm4
tf=3.77 cm	Wply=4243.000 cm3	Wplz=1945.000 cm3	

INTERNAL FORCES AND CAPACITIES:

N,Ed = 3440 kN	My,Ed = -420 kN*m	Mz,Ed = 110 kN*m	Vy,Ed = -26 kN
Nc,Rd = 8415 kN	My,pl,Rd = 1167 kN*m	Mz,pl,Rd = 535 kN*m	Vy,c,Rd = 3957 kN
Nb,Rd = 6640 kN	My,c,Rd = 1167 kN*m	Mz,c,Rd = 535 kN*m	Vz,Ed = -200 kN
	My,N,Rd = 774 kN*m	Mz,N,Rd = 503 kN*m	Vz,c,Rd = 1372 kN
	Mb,Rd = 1150 kN*m		
			Class of section = 1

LATERAL BUCKLING PARAMETERS:

z = 0.00	Mcr = 16778 kN*m	Curve,LT - a	XLT = 0.99
Lcr,low=4.20 m	Lam_LT = 0.26	fi,LT = 0.54	

BUCKLING PARAMETERS:

About Y axis:		About Z axis:	
Ly = 4.20 m	Lam_y = 0.23	Lz = 4.20 m	Lam_z = 0.59
Lcr,y = 2.94 m	Xy = 0.99	Lcr,z = 4.20 m	Xz = 0.79
Lamy = 20.30	kzy = 0.79	Lamz = 51.55	kzz = 0.78

VERIFICATION FORMULAS:

Section strength check:

N,Ed/Nc,Rd = 0.41 < 1.00 (6.2.4.(1))

My,Ed/My,c,Rd + Mz,Ed/Mz,c,Rd = 0.57 < 1.00 (6.2.5.(1))

(My,Ed/My,N,Rd)^ 2.00 + (Mz,Ed/Mz,N,Rd)^2.04 = 0.34 < 1.00 (6.2.9.1.(6))

Vy,Ed/Vy,c,Rd = 0.01 < 1.00 (6.2.6.(1))

Vz,Ed/Vz,c,Rd = 0.15 < 1.00 (6.2.6.(1))

Global stability check of member:

Lambda,y = 20.30 < Lambda,max = 210.00 Lambda,z = 51.55 < Lambda,max = 210.00 STABLE

My,Ed/Mb,Rd = 0.37 < 1.00 (6.3.2.1.(1))

N,Ed/(Xy*N,Rk/gM1) + kyy*My,Ed/(XLT*My,Rk/gM1) + kyz*Mz,Ed/(Mz,Rk/gM1) = 0.66 < 1.00 (6.3.3.(4))

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))

Section OK !!!

COMPARISON:

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

CONCRETE

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

VERIFICATION EXAMPLE 1 - Dimensioning reinforcement in rectangular section at bending

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

* - NOTE: the reference handbook [1] is based on the edition of Eurocode 2 EN1992-1-1:2004. The calculations in program Autodesk Robot Structural Analysis 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	
α_{cc} = 0.85		
Steel:	fyk=355	[MPa]

LOADS:

Bending moment M = 100kNm [cm^2]

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 α_{cc} = 0.85. The default value for the general edition of the code is α_{cc} = 1.0. In order to enable the comparison, change the factor to 0.85 in *Job Preferences/Design Codes/Partial factors for a Code EN 1992-1-1:2004 AC:2008/User defined (Fig.1.4).*

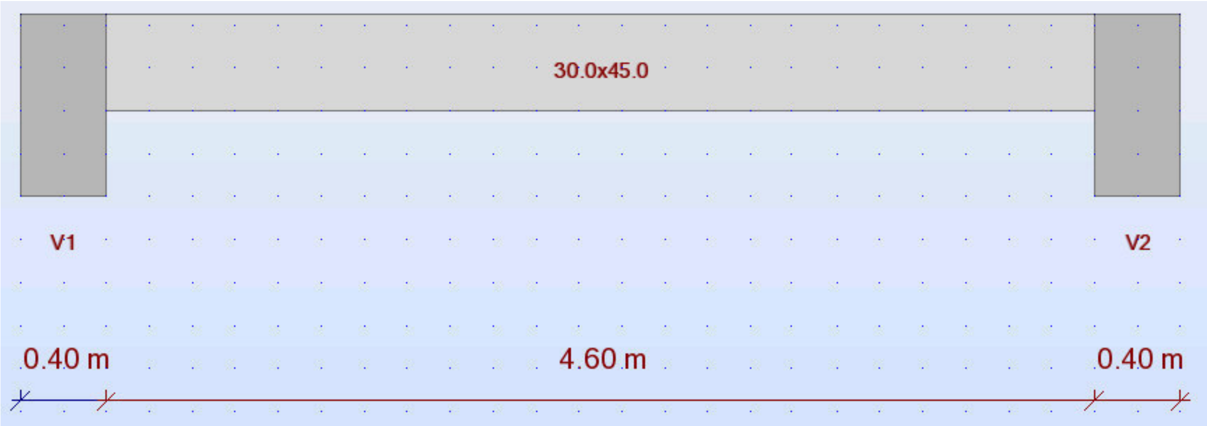
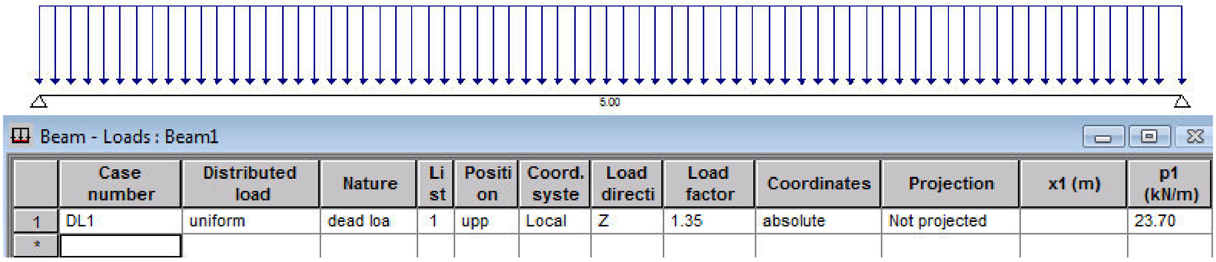


Fig. 1.1 Beam geometry



	Case number	Distributed load	Nature	List	Position	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

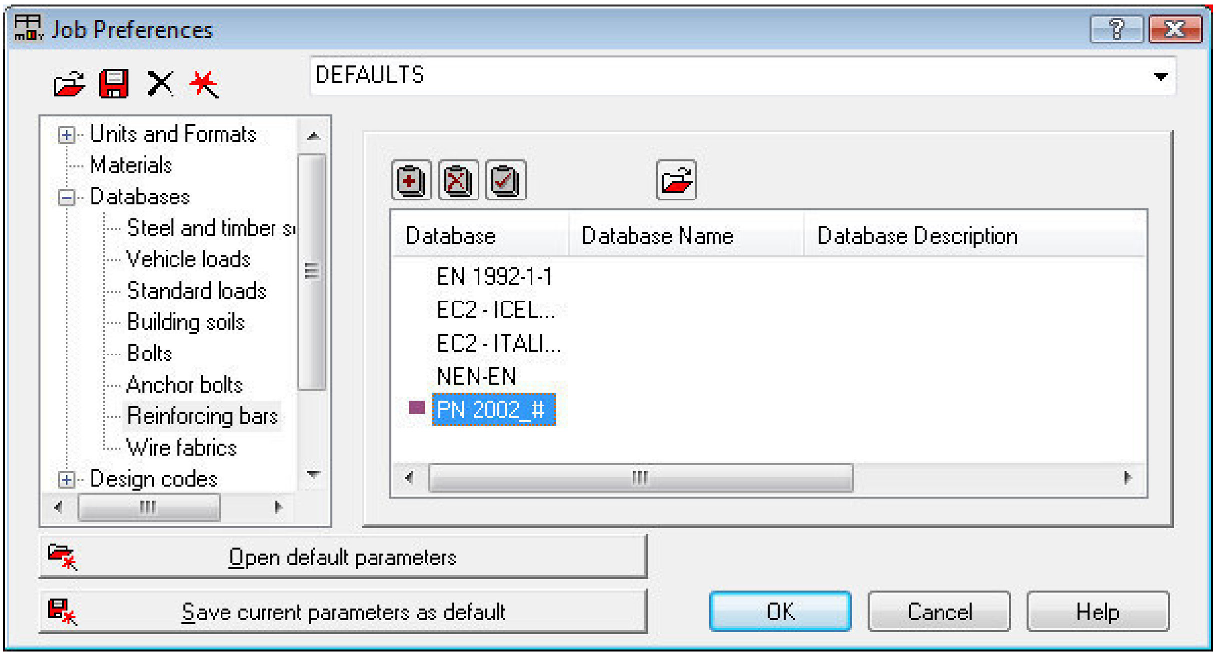


Fig. 1.3 Selection of steel database corresponding to [1]

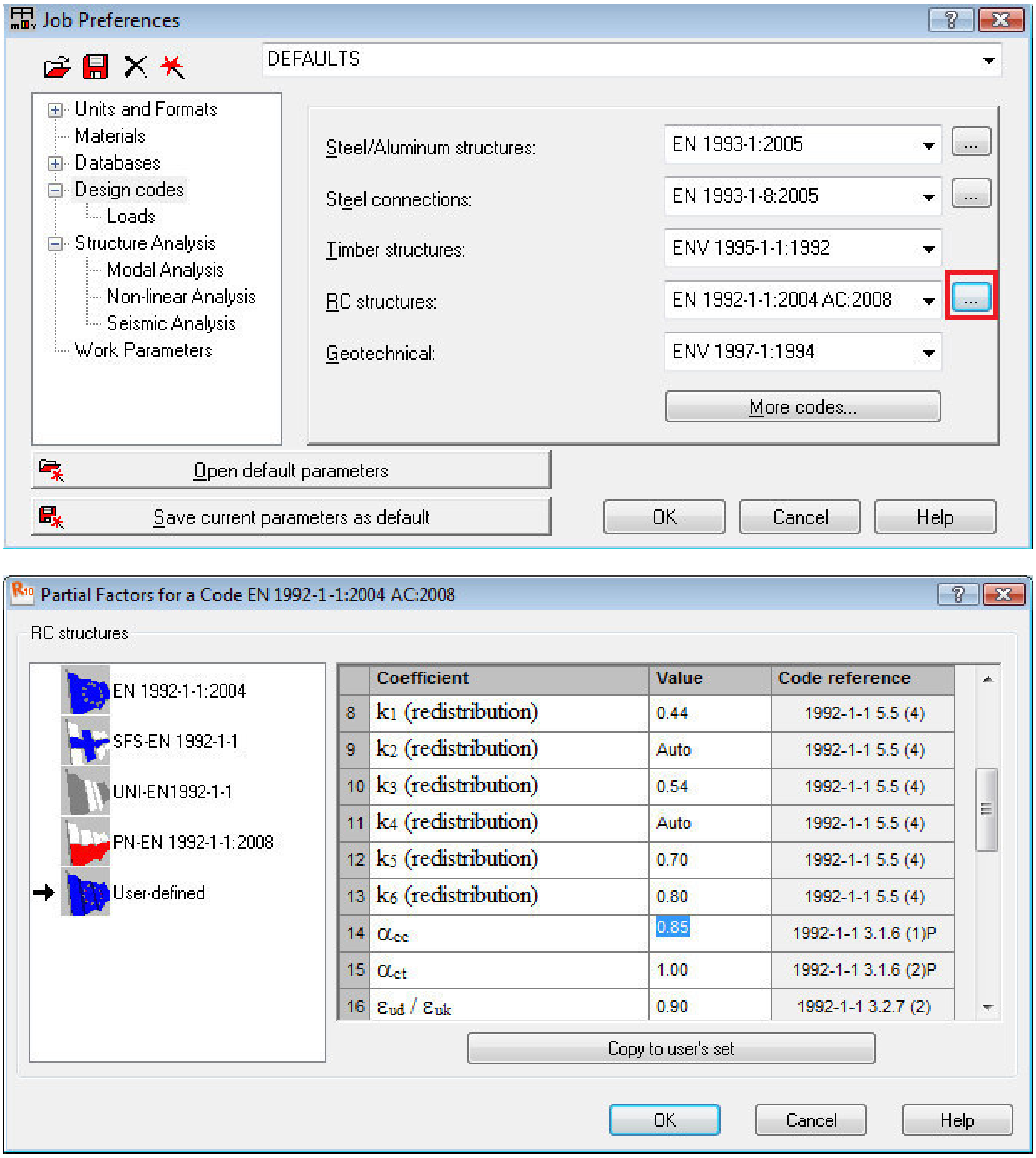


Fig. 1.4 Definition of partial factors

RESULTS OF LONGITUDINAL REINFORCEMENT (REINFORCEMENT FOR BENDING) CALCULATION:

The theoretical areas of reinforcement determined by the program are presented on the graph in *Fig.1.5.* The value in the midspan, compared with [1], is presented in the table below.

Theoretical areas	[1]	Robot
bottom reinf. A_{s1}	8.53 cm^2	8.57 cm^2

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

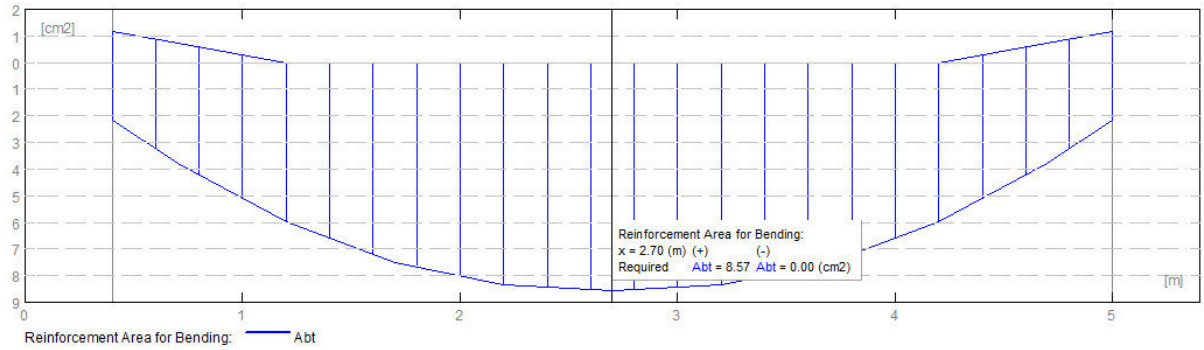


Fig. 1.5. Theoretical (required) areas of reinforcement in beam.

ANALYSIS OF RESULTS FOR NADs:

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

Code	γ_c	γ_s	α_{cc}	bottom reinf. A_{s1}-Robot results
Handbook example (general Eurocode 2 edition with modified α_{cc})	1.5	1.15	0.85	8.57 cm^2
EN 1992-1-1:2004 AC:2008	1.5	1.15	1.0	8.45 cm^2
PN-EN 1992-1-1:2008	1.4	1.15	1.0	8.41 cm^2
UNI-EN 1992-1-1	1.5	1.15	0.85	8.57 cm^2
SFS-EN 1992-1-1	1.5	1.15	0.85	8.57 cm^2
EN 1992-1-1 DK NA:2007	1.45	1.2	1.0	8.80 cm^2
BS EN1992-1-1:2004 NA2005	1.5	1.15	0.85	8.57 cm^2
NS-EN 1992-1-1:2004/NA:2008	1.5	1.15	0.85	8.57 cm^2
NF EN 1992-1-1/NA:2007	1.5	1.15	1.0	8.45 cm^2

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

VERIFICATION EXAMPLE 2 - Dimensioning reinforcement in rectangular section at bending

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

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

DESCRIPTION OF THE EXAMPLE:

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

RESULTS OF LONGITUDINAL REINFORCEMENT (REINFORCEMENT FOR BENDING) CALCULATION:

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

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

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

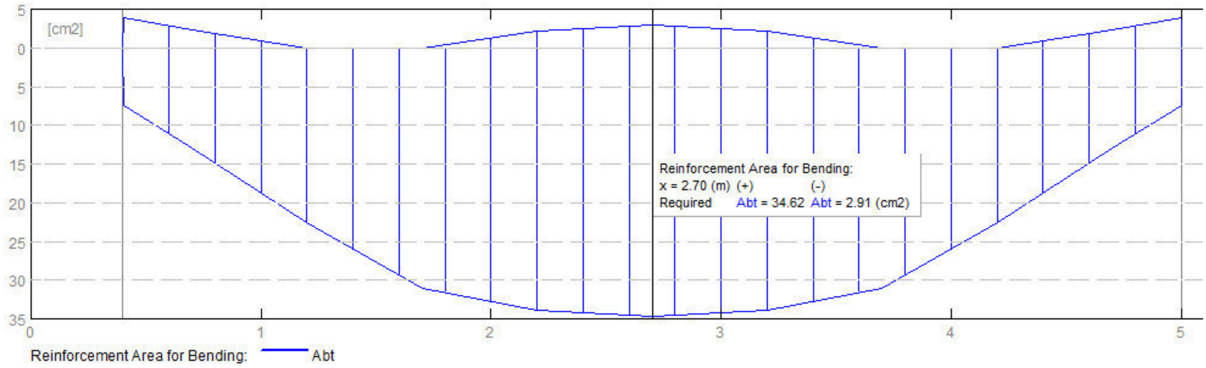


Fig. 2.1. Theoretical (required) areas of reinforcement in beam.

ANALYSIS OF RESULTS FOR NADs:

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

Code	γ_c	γ_s	α_{cc}	bottom reinf. A_{s1}- Robot results	top reinf. A_{s2}- Robot results
Handbook example (general Eurocode 2 edition with modified α_{cc})	1.5	1.15	0.85	34.73 cm^2	2.92 cm^2
EN 1992-1-1:2004 AC:2008	1.5	1.15	1.0	34.27 cm^2	0.0 cm^2
PN-EN 1992-1-1:2008	1.4	1.15	1.0	33.09 cm^2	0.0 cm^2
UNI-EN 1992-1-1	1.5	1.15	0.85	34.73 cm^2	2.92 cm^2
SFS-EN 1992-1-1	1.5	1.15	0.85	34.73 cm^2	2.92 cm^2
EN 1992-1-1 DK NA:2007	1.45	1.2	1.0	35.12 cm^2	0.0 cm^2
BS EN1992-1-1:2004 NA2005	1.5	1.15	0.85	34.73 cm^2	2.92 cm^2
NS-EN 1992-1-1:2004/NA:2008	1.5	1.15	0.85	34.73 cm^2	2.92 cm^2
NF EN 1992-1-1/NA:2007	1.5	1.15	1.0	34.27 cm^2	0.0 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 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_{s1}	11.62 cm^2	11.67 cm^2

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

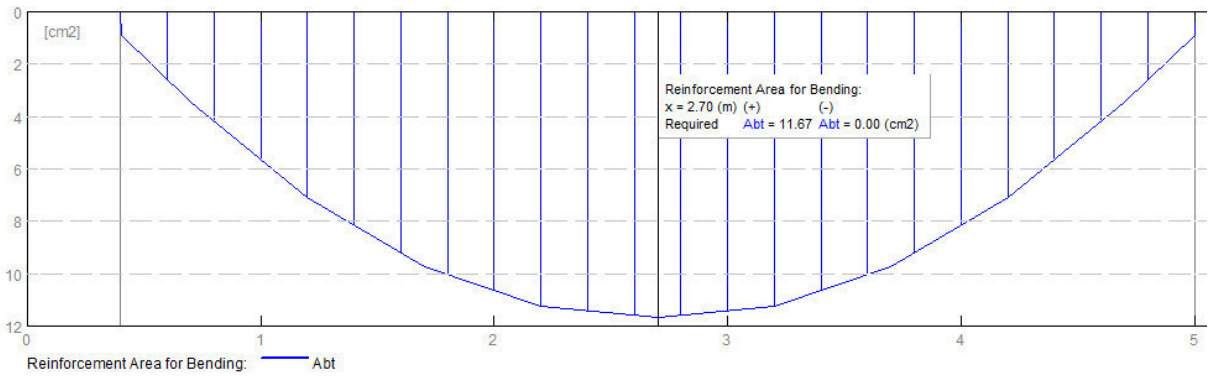


Fig. 3.1. Theoretical (required) areas of reinforcement in beam.

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 α_{cc} = 0.85. The default value for the general edition of the code is α_{cc} = 1.0. In this section, the same example is calculated for different national editions of Eurocode 2. The results of calculation are compared in the table below, along with the values of coefficients, which allows you to understand the possible differences for different NADs.

Code	γ_c	γ_s	α_{cc}	bottom reinf. A_{s1}-Robot results
Handbook example (general Eurocode 2 edition with modified α_{cc})	1.5	1.15	0.85	11.67 cm^2
EN 1992-1-1:2004 AC:2008	1.5	1.15	1.0	11.17 cm^2
PN-EN 1992-1-1:2008	1.4	1.15	1.0	11.00 cm^2
UNI-EN 1992-1-1	1.5	1.15	0.85	11.67 cm^2
SFS-EN 1992-1-1	1.5	1.15	0.85	11.67 cm^2
EN 1992-1-1 DK NA:2007	1.45	1.2	1.0	11.57 cm^2
BS EN1992-1-1:2004 NA2005	1.5	1.15	0.85	11.67 cm^2
NS-EN 1992-1-1:2004/NA:2008	1.5	1.15	0.85	11.67 cm^2
NF EN 1992-1-1/NA:2007	1.5	1.15	1.0	11.17 cm^2

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

VERIFICATION EXAMPLE 4
- Dimensioning reinforcement in rectangular section at bending with compression

Example based on:
[1] Bases of designing of reinforced and prestressed concrete structures according to Eurocode 2 (in Polish). Dolnośląskie Wydawnictwa Edukacyjne, Wroclaw 2006, Example 6.10, pp. 334 *

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

DESCRIPTION OF THE EXAMPLE:

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

RESULTS OF LONGITUDINAL REINFORCEMENT (REINFORCEMENT FOR BENDING) CALCULATION:

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

Theoretical areas	[1]	Robot
bottom reinf. A_{s1}	3.64 cm^2	3.64 cm^2
top reinf. A_{s2}	4.30 cm^2	4.34 cm^2

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

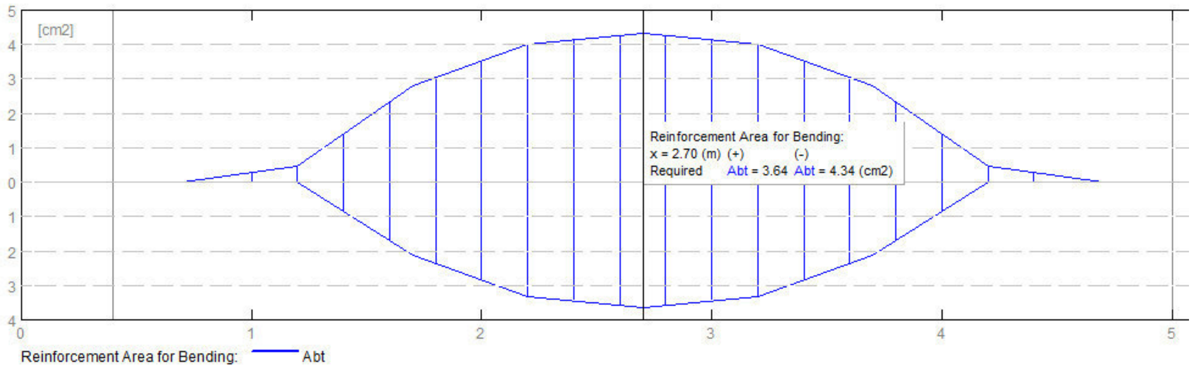


Fig. 4.1. Theoretical (required) areas of reinforcement in beam.

ANALYSIS OF RESULTS FOR NADs:

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

Code	γ_c	γ_s	α_{cc}	bottom reinf. A_{s1}- Robot results	top reinf. A_{s2}- Robot results
Handbook example (general Eurocode 2 edition with modified α_{cc})	1.5	1.15	0.85	3.64 cm^2	4.34 cm^2
EN 1992-1-1:2004 AC:2008	1.5	1.15	1.0	4.75 cm^2	0.0 cm^2
PN-EN 1992-1-1:2008	1.4	1.15	1.0	3.24 cm^2	0.0 cm^2
UNI-EN 1992-1-1	1.5	1.15	0.85	3.64 cm^2	4.34 cm^2
SFS-EN 1992-1-1	1.5	1.15	0.85	3.64 cm^2	4.34 cm^2
EN 1992-1-1 DK NA:2007	1.45	1.2	1.0	4.13 cm^2	0.0 cm^2
BS EN1992-1-1:2004 NA2005	1.5	1.15	0.85	3.64 cm^2	4.34 cm^2
NS-EN 1992-1-1:2004/NA:2008	1.5	1.15	0.85	3.64 cm^2	4.34 cm^2
NF EN 1992-1-1/NA:2007	1.5	1.15	1.0	4.75 cm^2	0.0 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 showthe results are correct for all cases.

VERIFICATION EXAMPLE 5 - Dimensioning of shear reinforcement in beam with rectangular section

Example based on:
Manual calculations according to:
[2] Eurocode 2 EN 1992-1-1:2004 AC:2008, point 6.2

DESCRIPTION OF THE EXAMPLE:

Calculate the shear reinforcement in simply supported beam with rectangular section. In this example, the results of the program are compared against the manual calculations presented.

GEOMETRY:

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

MATERIAL:

Concrete:	C20/255
Steel:	B500C (f_{yk} = 500 [MPa])

LOADS:

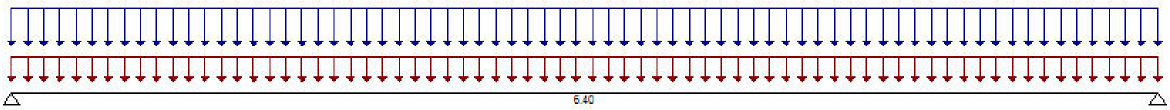
Uniformly distributed:

Dead load:	q_D = 30 [kN/m]
Live load:	q_L = 20 [kN/m]

IMPORTANT STEPS:

Define the geometry of the beam (*Fig.5.1*) and loads (*Fig.5.2*). Set proper concrete and steel in *Calculation Options*. Set allowable stirrups spacings to: 0.05; 0.07; 0.10; 0.20; 0.25; 0.30; 0.35; 0.40; 0.50.

Fig. 5.1 Beam geometry



Beam - Loads : Beam1

	Case number	Distributed load	Nature	List	Position	Coord. syste	Load directi	Load factor	Coordinates	Projection	x1 (m)	p1 (kN/m)
1	DL1	uniform	dead loa	1	upp	Local	Z	1.35	absolute	Not projected		30.00
2	LL1	uniform	Live	1	upp	Local	Z	1.50	absolute	Not projected		20.00
*												

Fig. 5.2 Loads and the calculation model

RESULTS OF SHEAR REINFORCEMENT DIMENSIONING:

- **CALCULATION OF MAXIMUM SHEAR FORCE:**

Load nature:	Characteristic load [kN/m]	Load factor	Design load [kN/m]
Dead load	30	1.35	40.5
Live load	20	1.5	30
		$q_{tot} =$	70.5

The shear force at the end of the beam is equal to:

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

$$l = 6.8m$$

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

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

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

- **CALCULATION OF SHEAR CAPACITY OF A BEAM WITHOUT SHEAR REINFORCEMENT:**

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

$$V_{Rd,c} = \left[C_{Rd,c} k (100 \rho_l f_{ck})^{1/3} + k_1 \sigma_{cp}\right] b_w d = 103.69kN$$

$$C_{Rd,c} = 0.18 / \gamma_c = 0.12$$

$$k = 1 + \sqrt{200/d} = 1.61 \leq 2.0$$

$$d = 600 - 65 = 535mm$$ (position of bottom bars is averaged for two layers)

$$\rho_l = \frac{A_{sl}}{b_w d} = 0.0117$$

$$A_{sl} = 2199mm^2$$

$$b_w = 350mm$$

$$f_{ck} = 20MPa$$

$$\sigma_{cp} = 0MPa$$

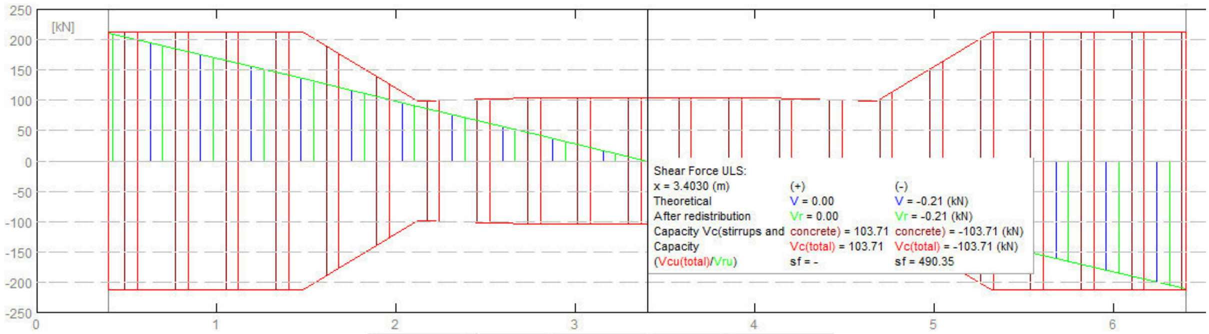
But should not be smaller than:

$$V_{Rd,c} = \left[v_{\min} + k_1 \sigma_{cp}\right] b_w d = 59.9kN$$

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

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 mid-span) (*Fig.5.3*).

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



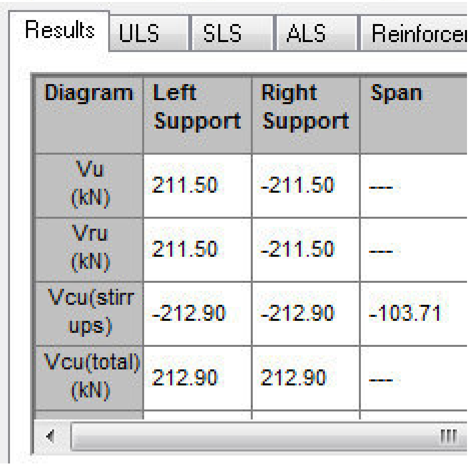


Diagram	Left Support	Right Support	Span
Vu (kN)	211.50	-211.50	---
Vru (kN)	211.50	-211.50	---
Vcu(stirrups)	-212.90	-212.90	-103.71
Vcu(total) (kN)	212.90	212.90	---

Fig. 5.3 Shear force distribution and shear capacity

- **CALCULATION OF SHEAR CAPACITY OF A BEAM WITH SHEAR REINFORCEMENT:**

Since, at the support face $V \geq V_{Rd,c}$ 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:

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

$$A_{sw} = 0.000101m^2$$ (2 bars ϕ8)

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

$$z = 0.9d = 0.49m$$

$$d = 0.6 - 0.059 = 0.541m$$ (for bottom bars at the support)

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

$$f_{ywk} = 500MPa$$

$$\gamma_s = 1.15$$

$$\cot\theta = 1.0$$ (set in Calcualtion options/General)

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

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

And should not be greater than: $VRd,\max = \dfrac{\alpha_{cw} b_w z \nu_1 f_{cd}}{\cot\theta + \tan\theta} = 627.4kN$

$$\alpha_{cw} = 1.0$$

$$\nu_1 = 0.552$$

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

The value of $V_{Rd,s}$ at the support face calculated by the program (*Fig.5.3*) is in agreement with the one calculated above (see table below).

	Manual calculation	Robot
Shear capacity $V_{Rd,s}$	212.9 kN	212.9 kN

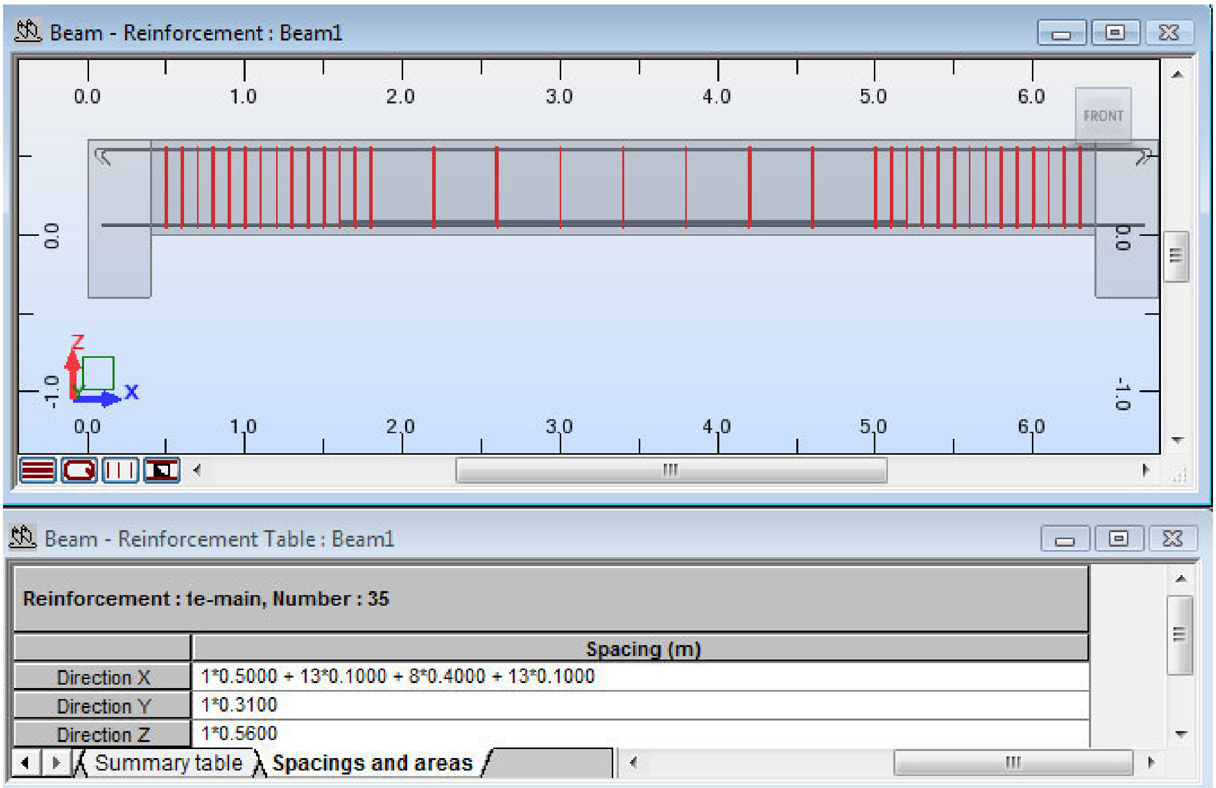


Fig. 5.4 Shear reinforcement distribution (see Direction X in the Reinforcement table)

ANALYSIS OF RESULTS FOR NADs:

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

Code	γ_c	γ_s	α_{cc}	Shear capacity $V_{Rd,c}$	Shear capacity $V_{Rd,s}$
EN 1992-1-1:2004 AC:2008 (manual calculation)	1.5	1.15	1.0	103.71 kN	212.9 kN
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 NA2005	1.5	1.15	0.85	103.71 kN	236.03 kN
NS-EN 1992-1-1:2004/NA:2008	1.5	1.15	0.85	103.71*	236.03 kN
NF EN 1992-1-1/NA:2007	1.5	1.15	1.0	103.71 kN	212.13 kN

As it can be seen, the value of shear capacity $V_{Rd,s}$ is dependent upon the varying γ_c coefficient for different national editions of the code. The difference concerning the value of $V_{Rd,c}$ is due to the $C_{Rd,c}$ coefficient dependent upon γ_c.

* NOTE: The spacing of of stirrups of 40cm used in other editions of the code is greater than the maximum allowable spacing according to NS-EN 1992-1-1:2004/NA:2008, thus the spacing of stirrups in the mid-span should be decreased down to 25cm.

VERIFICATION EXAMPLE 6 - Deflection of simply supported beam with rectangular section

Example based on:
[1] Bases of designing of reinforced and prestressed concrete structures according to Eurocode 2 (in Polish). Dolnośląskie Wydawnictwa Edukacyjne, Wroclaw 2006, Example 11.9.5, pp. 642 *

* - NOTE: the reference handbook [1] is based on the edition of Eurocode 2 EN1992-1-1:2004. The calculations in program Autodesk Robot Structural Analysis 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: 5ϕ20

LOADS:

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

IMPORTANT STEPS:

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

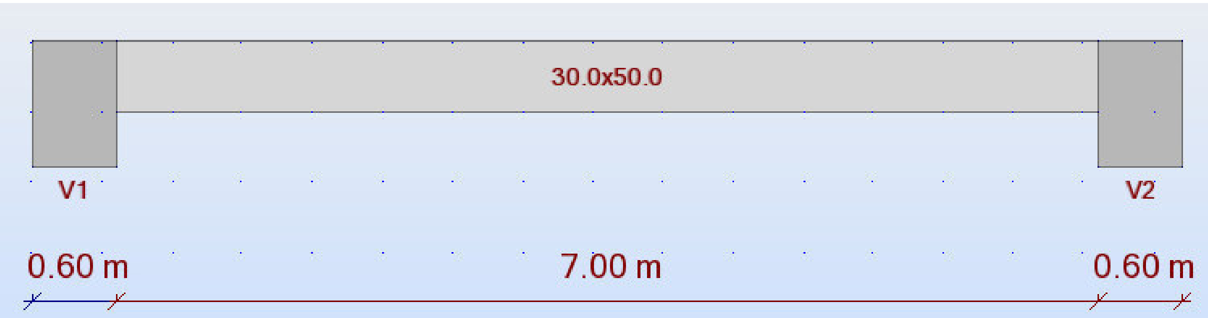
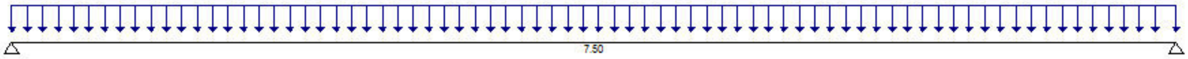


Fig. 6.1 Beam geometry



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

Fig. 5.2 Loads and the calculation model

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

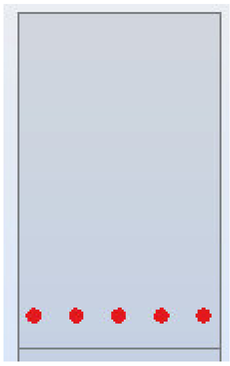


Fig. 5.2 Reinforcement (5ϕ20) assumed in [1]

RESULTS OF DEFLECTION CALCULATION:

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

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

$$\xi = 0.9686$$

$$f_I = 2.720 cm$$

$$f_{II} = 3.791 cm$$

	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 $E_{cm} = 27500 MPa$ while Robot uses the code value for C16/20 concrete, $E_{cm} = 29000 MPa$.

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.

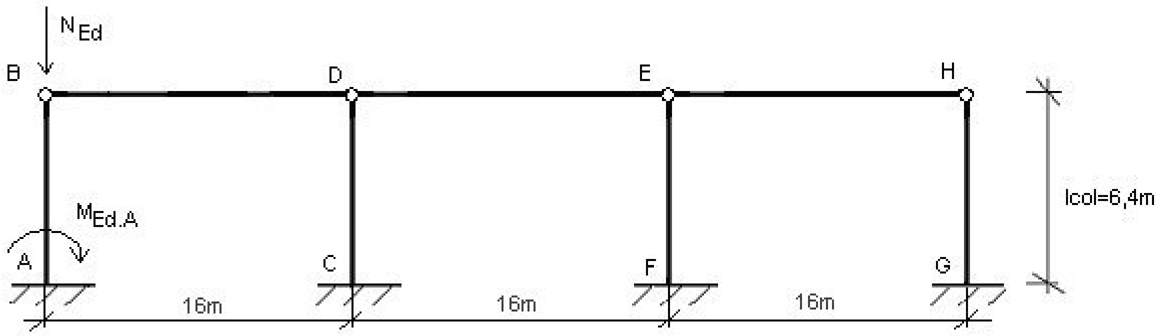


Fig. 1.1. The model of the frame with the analyzed column AB.

GEOMETRY:

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

MATERIAL:

Concrete: C30/37

α_{cc} = 0.85

Creep coefficient: φ = 2.3

Steel: fyk=410 [MPa]

LOADS:

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

REINFORCEMENT:

5 bars ϕ20 at both sides of the section (*Fig.1.9*)

IMPORTANT STEPS:
Define the geometry of the column and the buckling model in *Buckling length* dialog (*Fig.1.2*). The direction considered is direction Y (the unidirectional bending option will be enabled in next steps).

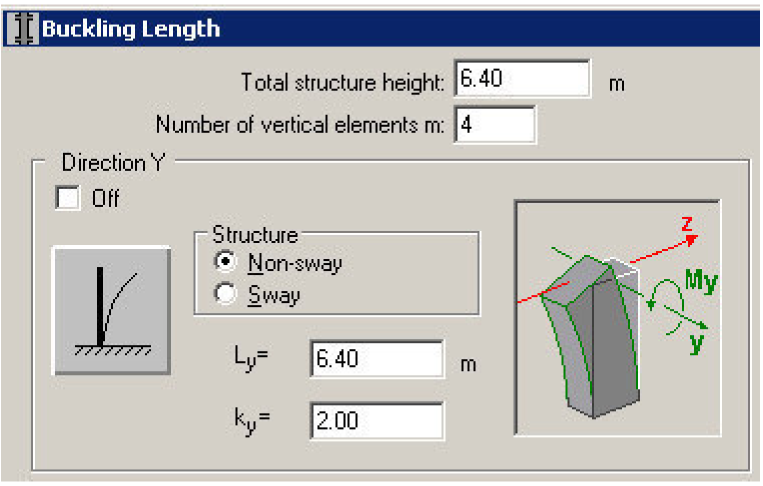


Fig. 1.2 Buckling parameters

Define the loads *(Fig.1.3)* and the parameter M_{0Eqp}/M_{0Ed} (ratio of quasi permanent moment to toal moment) – denoted in load table as Nd/N.

No.	Case	Nature	Group	N (kN)	MyA (kN*m)	MyB (kN*m)	MyC (kN*m)	MzA (kN*m)	MzB (kN*m)	MzC (kN*m)	Nd/N	γ
1	DSGN1	design	1	776.00	0.00	168.00	100.80	0.00	0.00	0.00	0.60	1.00

Fig. 1.3 Loads

Set creep coefficient as fixed value in *Story parameters* dialog.
Set proper concrete and steel with fyk=410MPa (34GS) in *Calculation Options*. In order to select steel different than available by default for EN1992-1-1 code (i.e. with fyk=410MPa) which is used in [1], select PN_2002# database in *Job Preferences/Databases/Reinforcing bars (Fig.1.4)*.

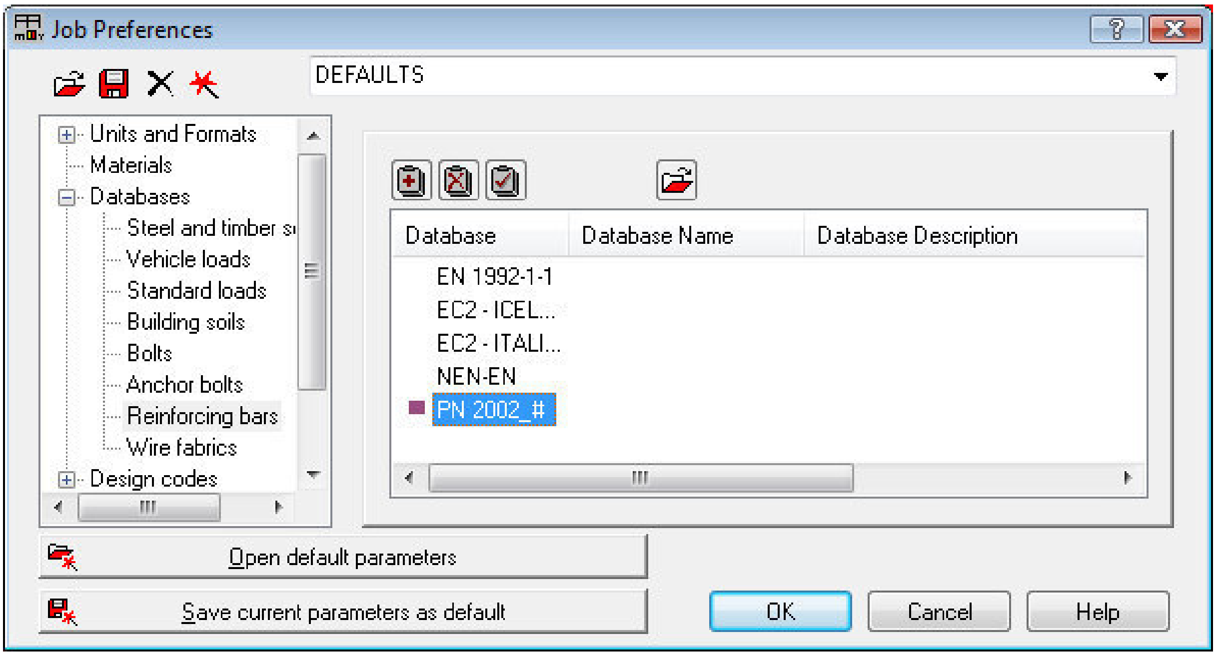


Fig. 1.4 Selection of steel database corresponding to [1]

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

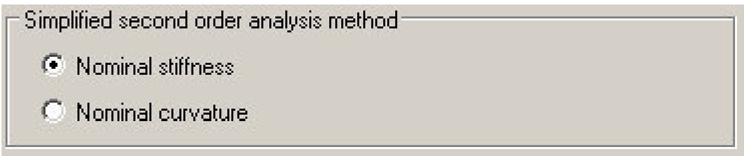


Fig. 1.5 Selection of second order analysis method

In order to enable unidirectional bending analysis, select "Design for simple bending" in *Calculation options/General* dialog (*Fig.1.6*).

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*).

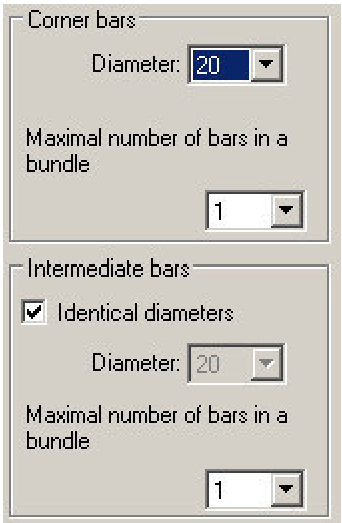
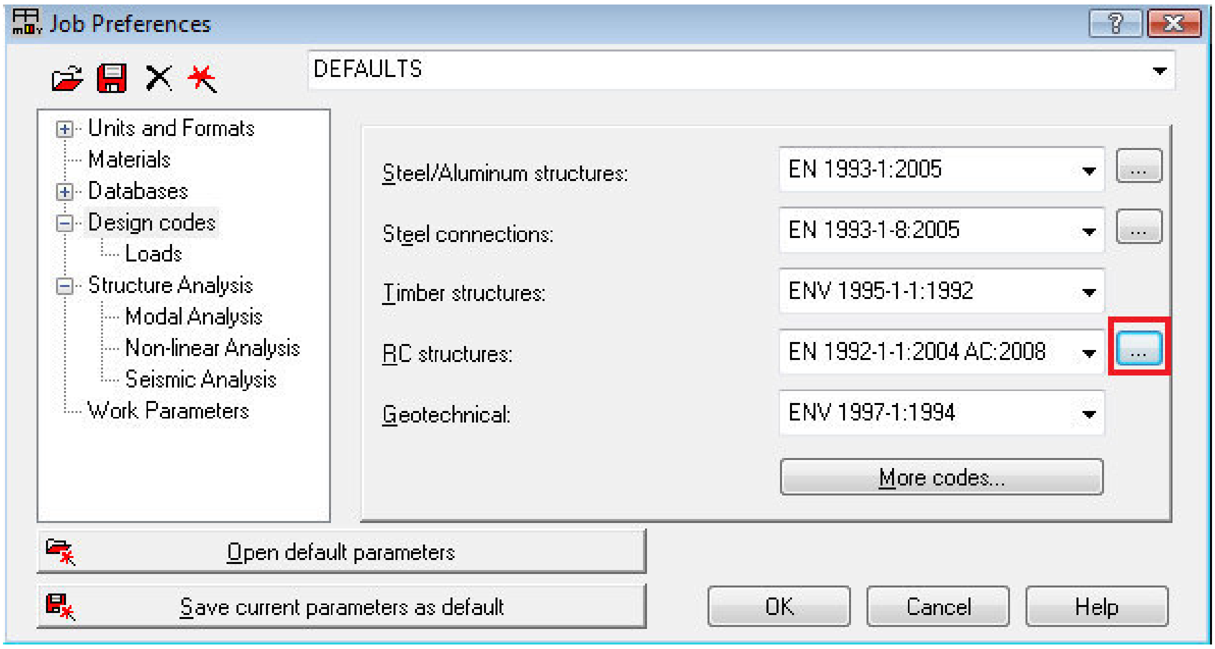


Fig. 1.7 Parameters of reinforcement

The authors of [1] use the partial factor α_{cc} = 0.85. The default value for the general edition of the code is α_{cc} = 1.0. In order to enable the comparison, change the factor to 0.85 in *Job Preferences/Design Codes/Partial factors for a Code EN 1992-1-1:2004 AC:2008/User defined (Fig.1.4).*



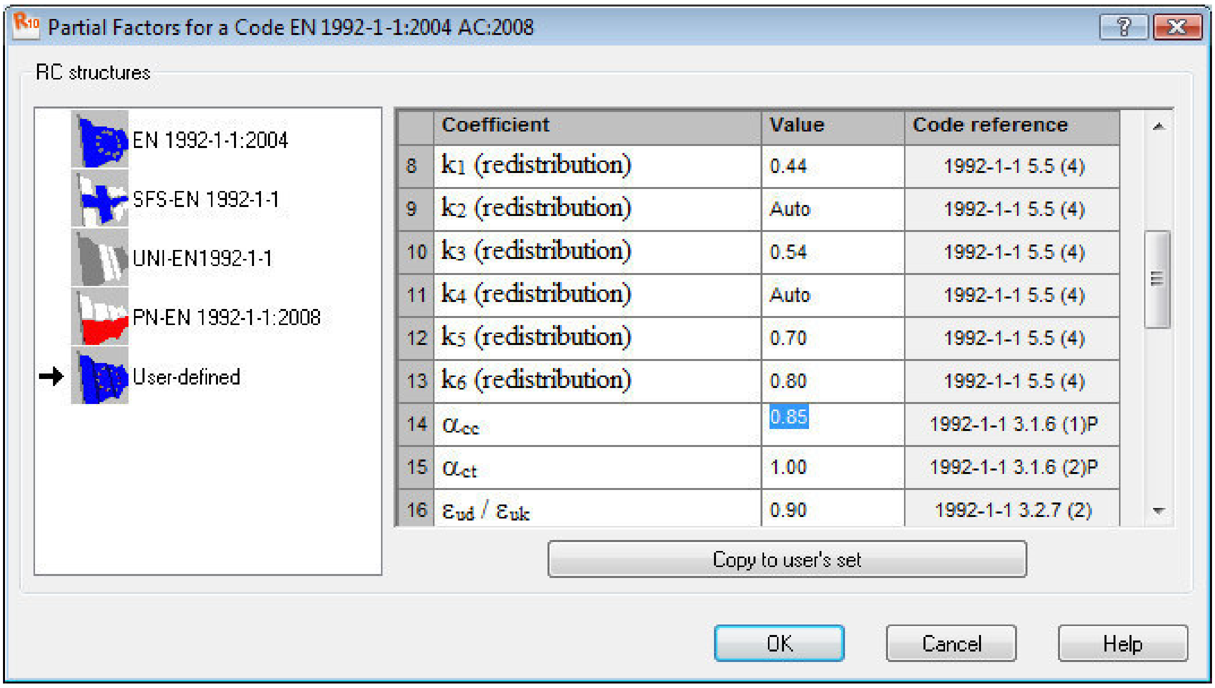


Fig. 1.8 Definition of partial factors

NOTE: The program automatically generates smaller reinforcement (8 ϕ20 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*)

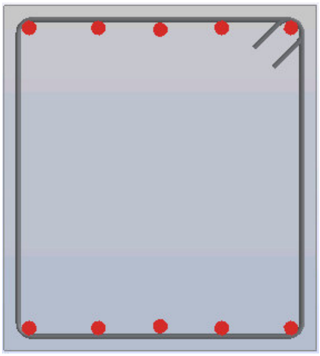


Fig. 1.9 Reinforcement assumed for the calculation (10 ϕ20).

RESULTS OF BUCKLING ANALYSIS - NOMINAL CURVATURE METHOD:

	(Unit)	[1]	Robot (results presented in calculation note)
$\lambda_{\lim}$	(-)	32.2	32.3
α_h	(-)	0.791	0.791
α_m	(-)	0.791	0.791
e_a	(cm)	2.0	2.0
K_r	(-)	1.0	1.0
K_φ	(-)	1.0	1.0
$1/r_0$	(1/m)	0.00863	0.00853*
$1/r_0$	(1/m)	0.00863	0.00853
c	(-)	10	10
e_2	(cm)	13.7 (14.1)**	14.0
M_{Ed}	(kNm)	289.8 (293.9)**	291.97

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

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

ANALYSIS OF RESULTS FOR NADs:

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

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

RESULTS OF BUCKLING ANALYSIS - NOMINAL STIFFNESS METHOD:

	(Unit)	[1]	Robot (results presented in calculation note)
J_s	(cm^4)	14500	14442
J_c	(cm^4)	785000**	468750
EJ	(kNm^2)	38670**	34285
N_b	(kN)	2330	2065
β	(-)	$\pi^2/12 = 0.8225$	$\pi^2/8 = 1.2337$***
M_{Ed}	(kNm)	258.9	319.79***

NOTES ON DIFFERENCES IN THE COMPARISON ABOVE

** - apparently, the calculation error was made in [1]. The Robot gives proper value of J_c.

*** - the authors of [1] take the value of c_0 = 12 for triangular distribution of moment. In Robot program however, this value is by default assumed as c_0 = 8 since the exact distribution of moment along the height of the column is not known (thus, more unfavorable case is chosen). Thus, β is taken as $\pi^2/8 = 1.2337$ when the moment in the mid-height (Mc) is not fixed by the user in the load definition dialog and $\beta = 1$ is assumed when Mc is fixed (i.e. when neither 5.8.7.3 (2) nor (3) can be applied). It naturally leads to the greater (in this particular case by 20%), but at the same time safer, value of total moment.

ANALYSIS OF RESULTS FOR NADs:

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

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

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	[cm^2/m]
reinforcement ratio:	ρ_x=ρ_y=0.0077	

MATERIAL:

Concrete:	fck = 15	[MPa]

IMPORTANT STEPS:

In the Structure model/Geometry view, define the slab with the supporting column in the middle. The slab 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 send the slab to provided RC calculations. On the *Slab-provided reinforcement* view, in *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.

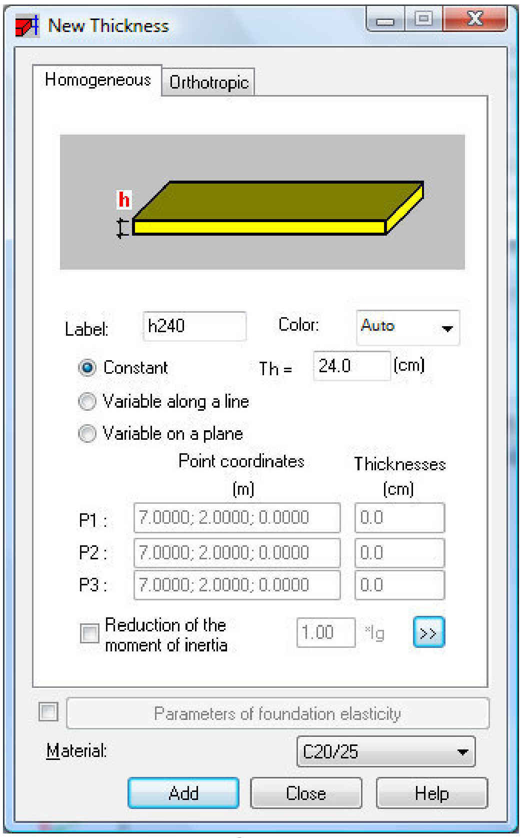


Fig. 1.1 Slab thicknes

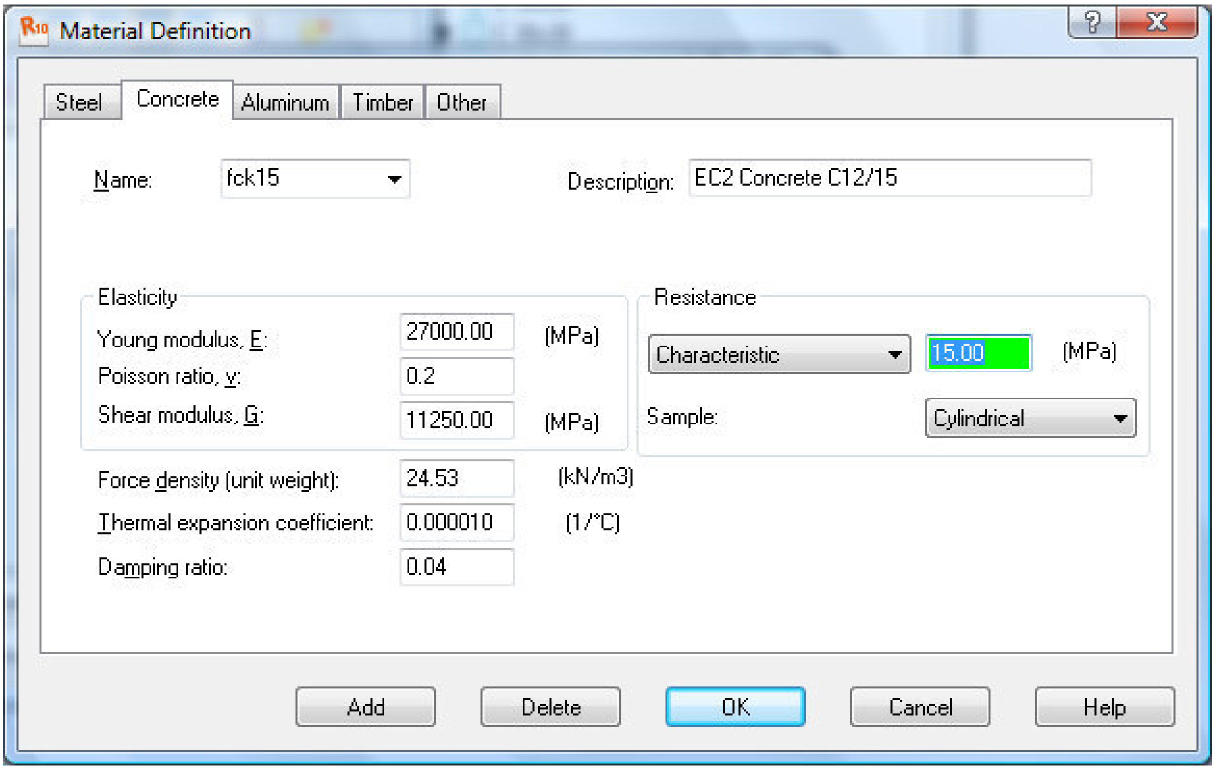


Fig. 1.2 Definition of new concrete type

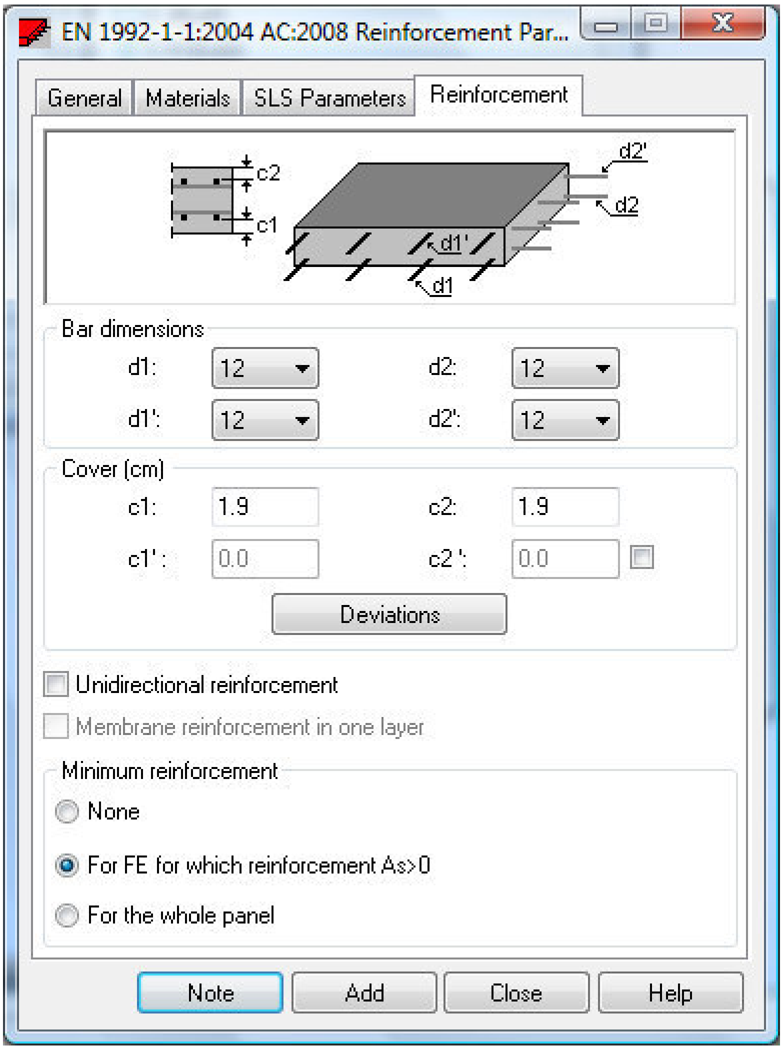


Fig. 1.3 Definition of covers of reinforcement

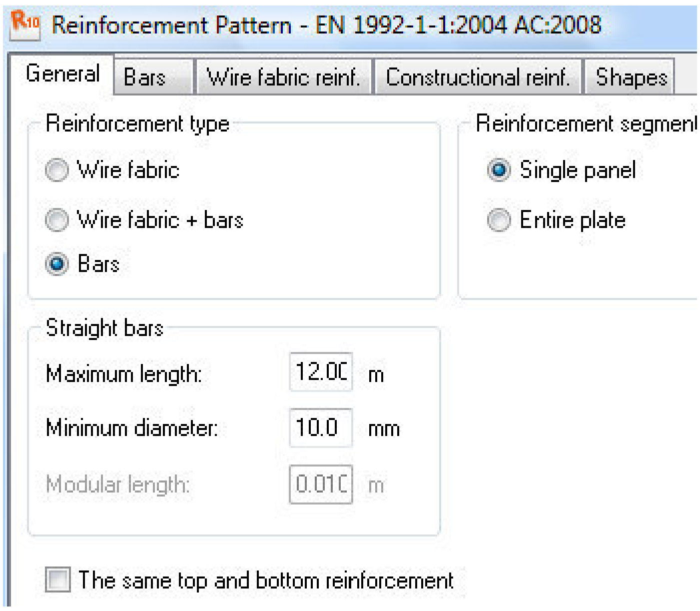


Fig. 1.4 Selection of reinforcement with bars

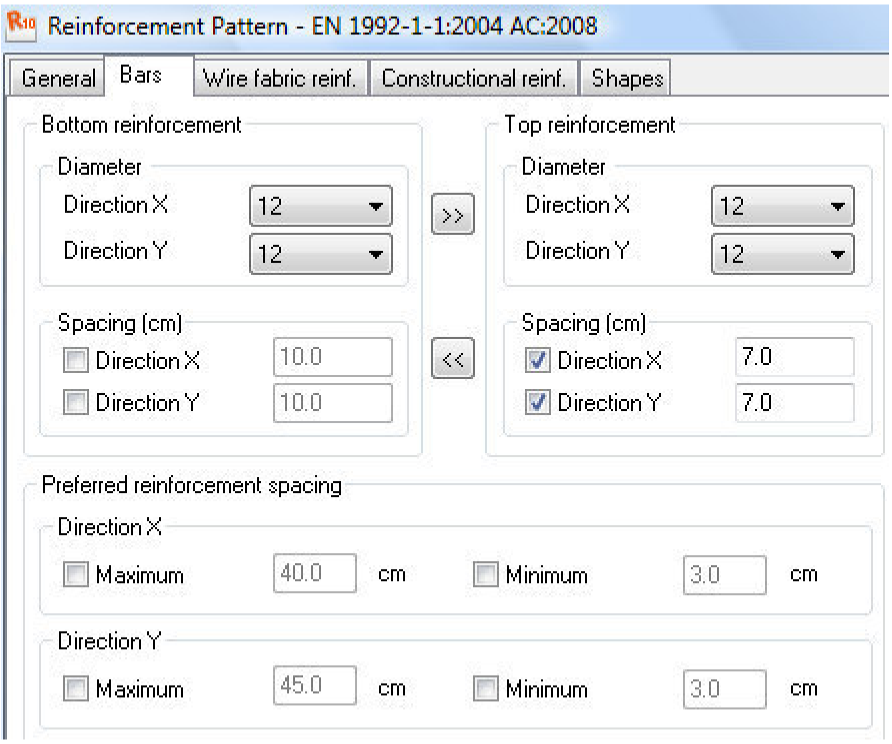


Fig. 1.5 Definition of spacing and diameters of reinforcement

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.

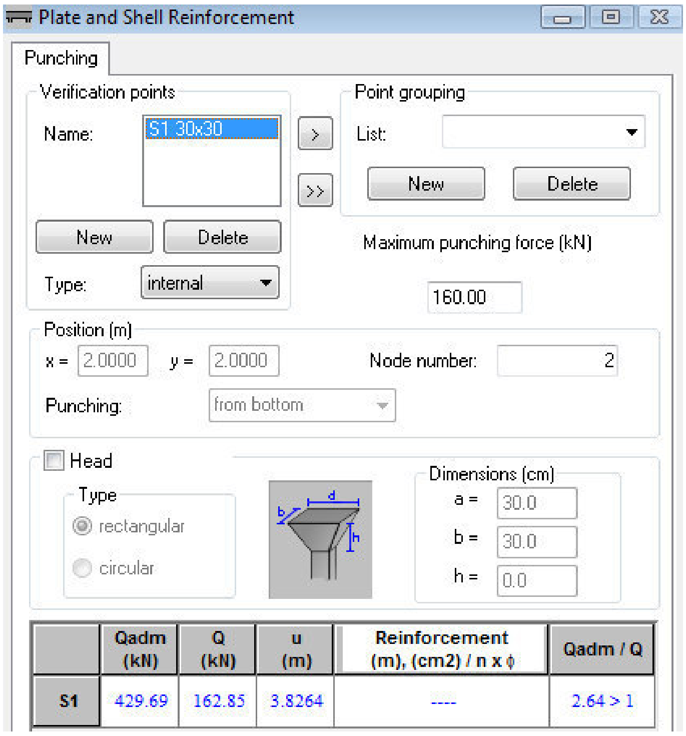


Fig. 1.6. Punching calculations dialog.

ANALYSIS OF RESULTS FOR NADs:

The presented example has been calculated for the general edition of Eurocode 2 [2]. In this section, the same example is calculated for different national editions of the Eurocode 2. The results of calculation are compared in the table below, along with the values of partial coefficients which allows you to tounderstand the possible differences for different NADs.

Code	γ_c	Punching capacity
EN 1992-1-1:2004 AC:2008	1.5	430 kN
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 NA2005	1.5	430 kN
NS-EN 1992-1-1:2004/NA:2008	1.5	430 kN
NF EN 1992-1-1/NA:2007	1.5	457 kN

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

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

Example based on:
Manual calculation

DESCRIPTION OF THE EXAMPLE:

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

GEOMETRY:

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

REINFORCEMENT:

reinforcement area:	A_x=A_y=16.08 [cm^2]
reinforcement ratio:	ρ_x=ρ_y=0.0077

MATERIAL:

Concrete: C20/25

FORCES IN THE NODE:

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

CALCULATION OF PUNCHING CAPACITY:

$$V_c = k\beta(1+50\rho)udf_{ctd} = 210kN \qquad (2.38)$$

$$k = 1.6 - d[m] = 1.391 \qquad (\rho c = 2500\ kg/m^3)$$

$$d = 0.209m$$

$$\rho = 0.0077$$

$$u = 2(c_x + d + c_y + d) = 2.036m$$

$$c_x = c_y = 0.3m$$

$$f_{ctd} = f_{ctk} / \gamma_c = 1.0MPa$$

$$f_{ctk} = 1.5MPa$$

$$\gamma_c = 1.5$$

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

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

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

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

$$A_u = 0.426m^2$$

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

	Manual calculation	Robot
Punching capacity	211 kN	211 kN

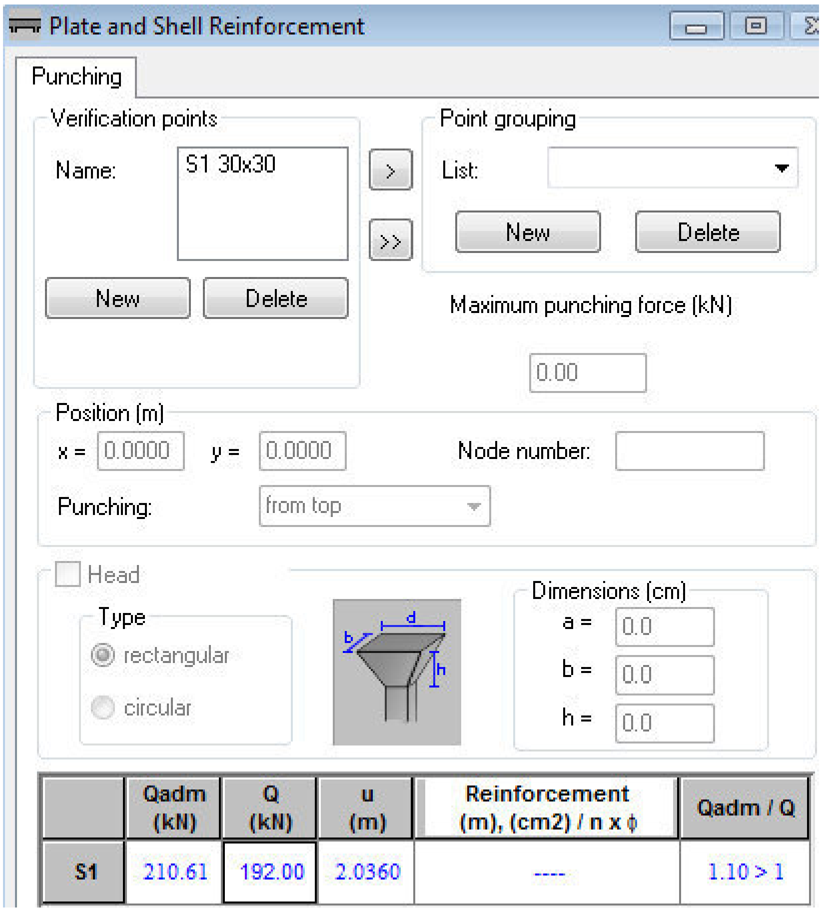


Fig. 2.1. Punching calculations dialog.

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

Example based on:
Manual calculation

DESCRIPTION OF THE EXAMPLE:

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

GEOMETRY:

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

REINFORCEMENT:

reinforcement area:	A_x=A_y=16.08	[cm^2]
reinforcement ratio:	ρ_x=ρ_y=0.0077	

MATERIAL:

Concrete: C20/25

FORCES IN THE NODE:

Vertical reaction:	V = 192 kN
Moments:	M_x = 24 kN
	M_y = 40 kN

CALCULATION OF β COEFFICIENT:

In Robot, β coefficient is calculated for both directions according to the equation (6.38) [2] modified for biaxial bending into a form:

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

$$u = 4.2264m$$

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

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

$$W_x = 0.5{c_y}^2 + c_y c_x + 4c_x d + 16d^2 + 2\pi d c_y = 1.706$$

$$W_y = 0.5{c_x}^2 + c_x c_y + 4c_y d + 16d^2 + 2\pi d c_x = 1.881$$

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

$$Q = v_{Ed} \cdot A_u = 342kN$$

$$A_u = ud = 0.883m^2$$

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	Robot
Punching force	342 kN	345 kN

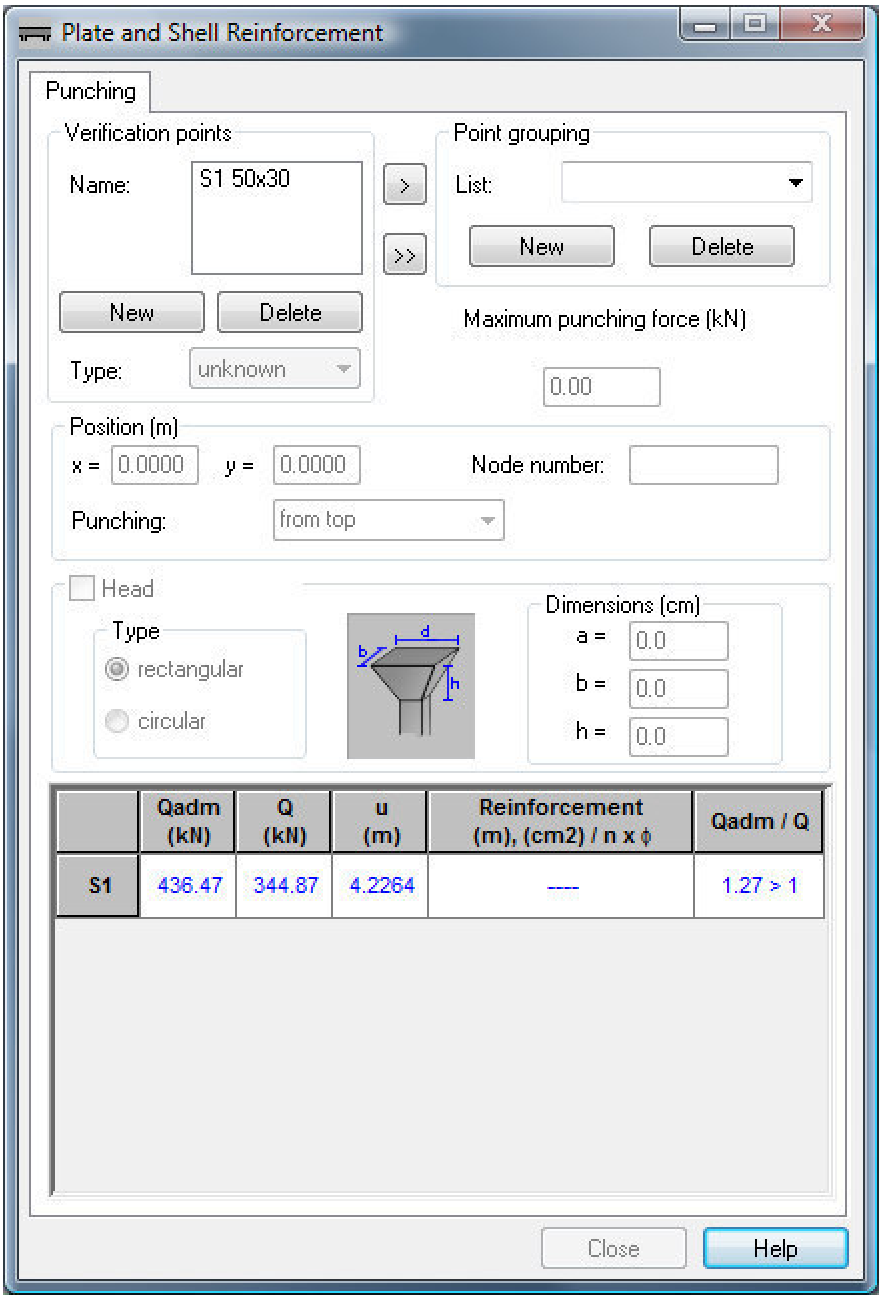


Fig. 3.1. Punching calculations dialog.

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 AC:2008	345 kN
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 NA2005	345 kN
NS-EN 1992-1-1:2004/NA:2008	345 kN
NF EN 1992-1-1/NA:2007	345 kN

As it can be seen, the results for different NADs are equal.

VERIFICATION PROBLEM 4 - Punching capacity of slab with shear reinforcement

Example based on:
[1] Bases of designing of reinforced and prestressed concrete structures according to Eurocode 2 (in Polish). Dolnośląskie Wydawnictwa Edukacyjne, Wroclaw 2006, Example 9.6.1, pp. 501 *

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

DESCRIPTION OF THE EXAMPLE:

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

GEOMETRY:

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

REINFORCEMENT:

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

MATERIAL:

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

LOADS:

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

IMPORTANT STEPS:

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

NOTE:

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

Having calculated the strucutre model and the RC required reinforcement, send the slab to provided RC calculations. On the *Slab-provided reinforcement* view, in *Reinforcement pattern/General* dialog select reinforcement with bars. On the *Bars* tab (*Fig.4.4*), set diameters to 12mm, and the spacing of top bars to 7cm (in order to obtain the reinforcement ratio as in Handbook example). Now, the calculations of real reinforcement, along with punching calculations may be carried out.

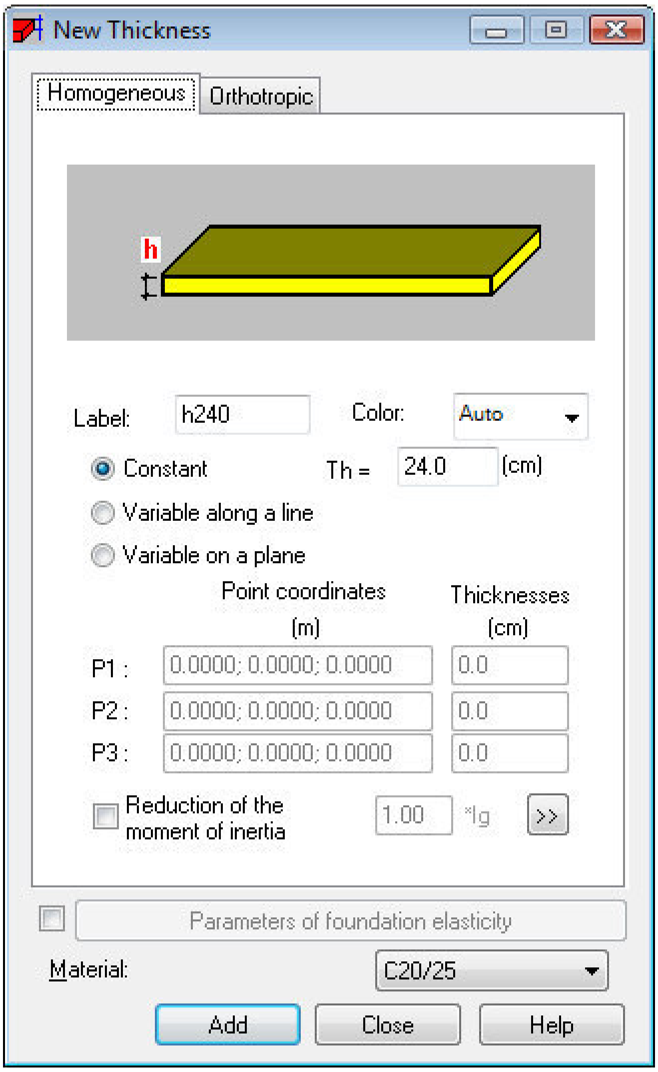


Fig. 4.1. Slab thickness dialog

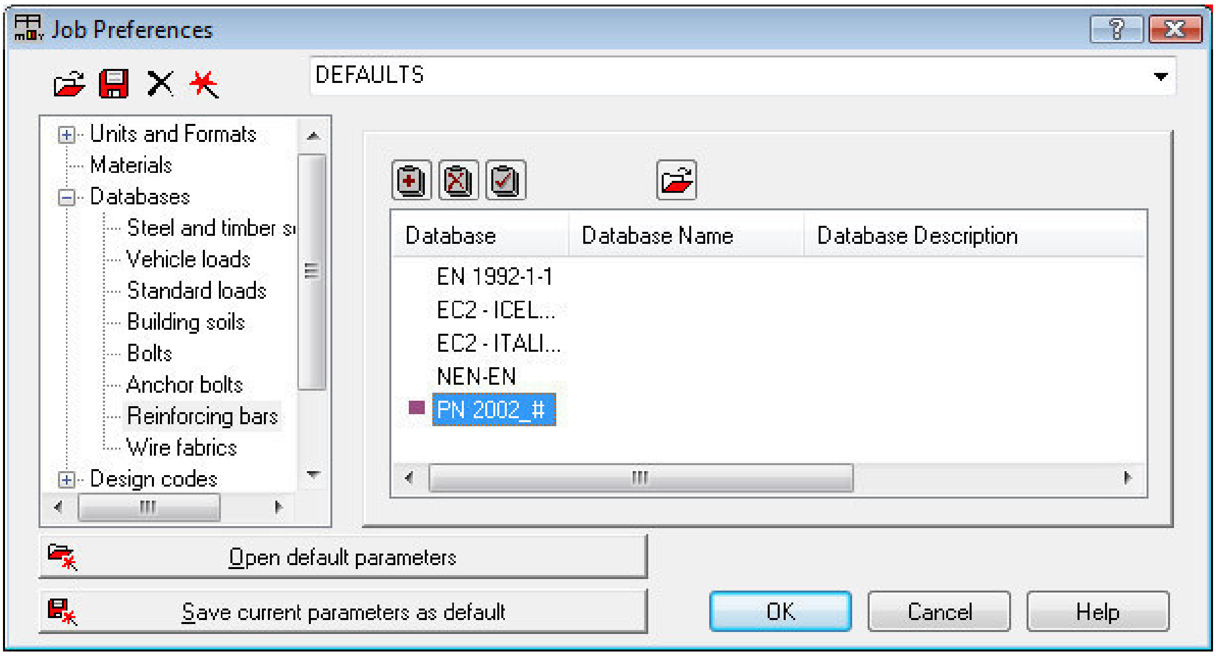


Fig. 4.2 Selection of steel database corresponding to [1]

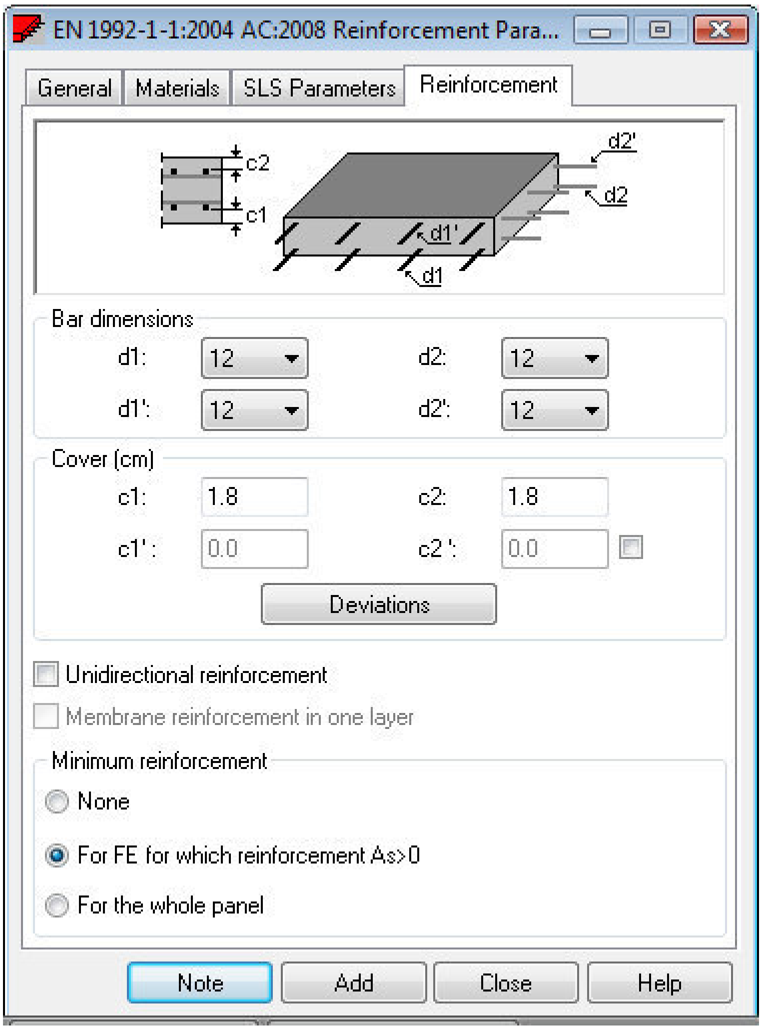


Fig. 4.3 Definition of covers of reinforcement

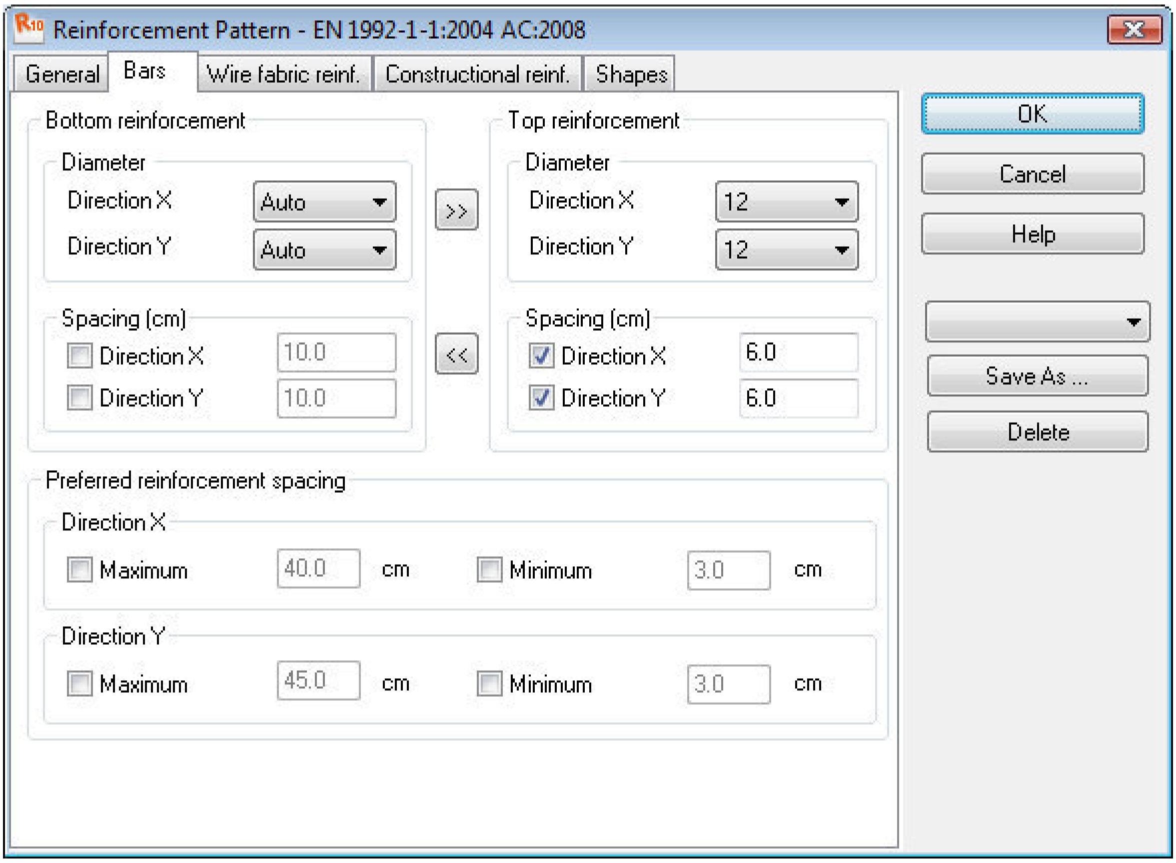


Fig. 4.4 Definition of spacing and diameters of reinforcement

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^2, while in Robot it is 4.14 cm^2 (see table below). This relatively small difference results from the assumed spacing of perimeters assumed during calculation of theoretical reinforcement. In Robot, the spacing is assumed as eual to the maximum allowable value s_r=0.75d, while in [1], the assumed value is smaller than this maximum.

	[1]	Robot
Punching reinforcement	2 perimeters A=3.96 cm^2	2 perimeters A=4.14 cm^2

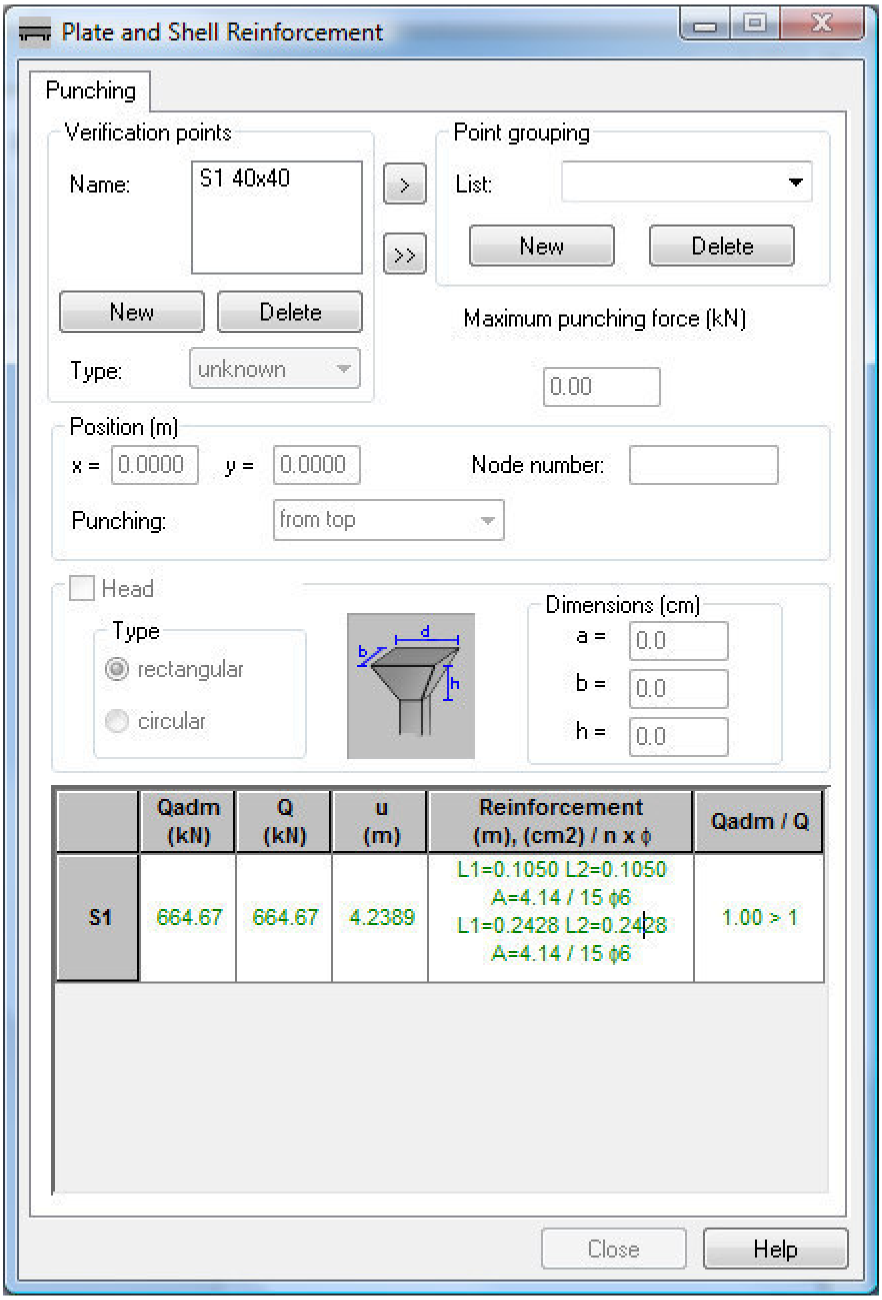


Fig. 4.5. Punching calculations dialog.

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

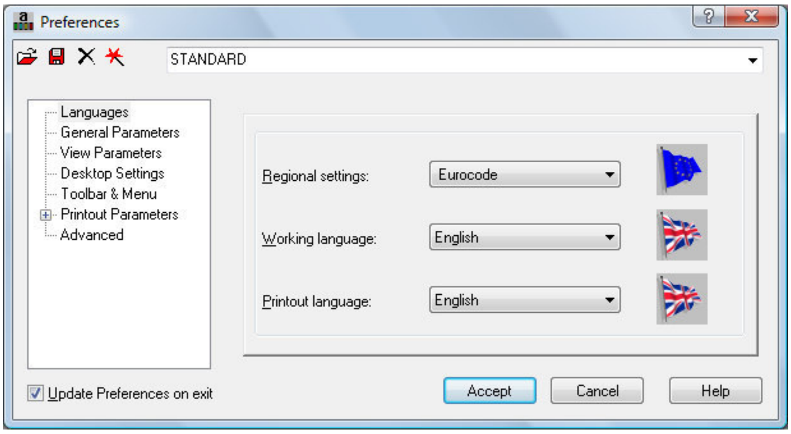
EN 1995-1:2004/A1:2008, March, 2005

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).

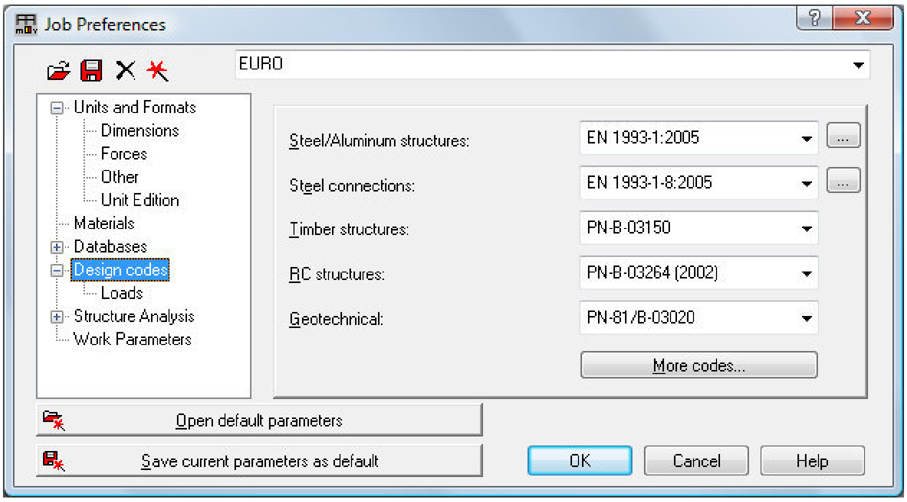
A. Preferences

To specify your regional preferences in PREFERENCES dialog click Tools/ Preferences.
Default PREFERENCES dialog opens e.g.:

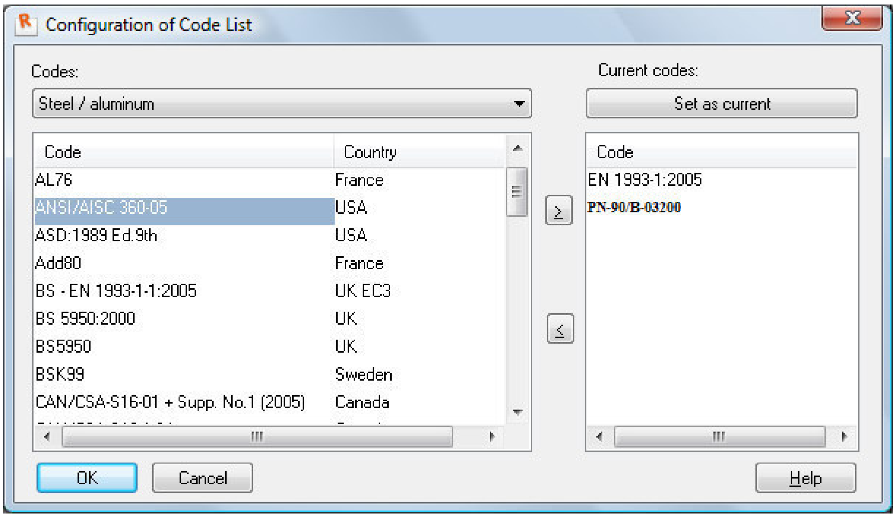


B. Job Preferences

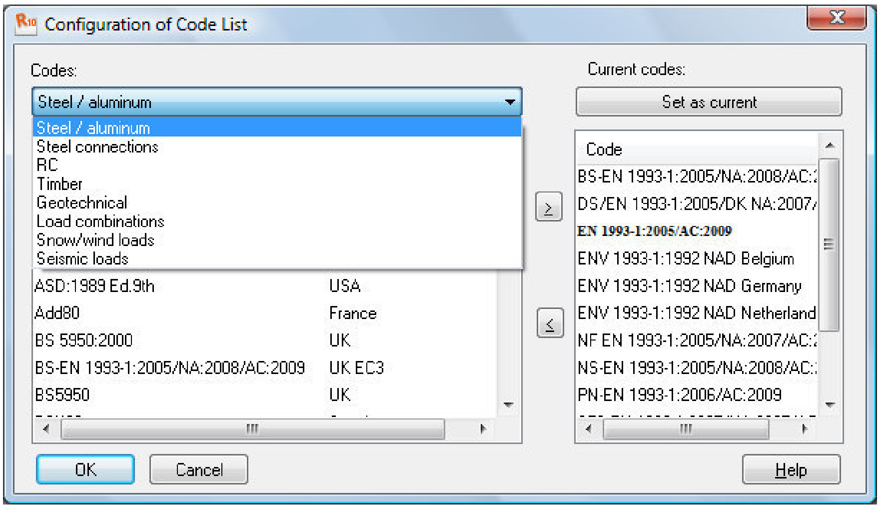
To specify your job preferences in JOB PREFERENCES dialog click Tools/ Job Preferences.
Default JOB PREFERENCES dialog opens, e.g.:



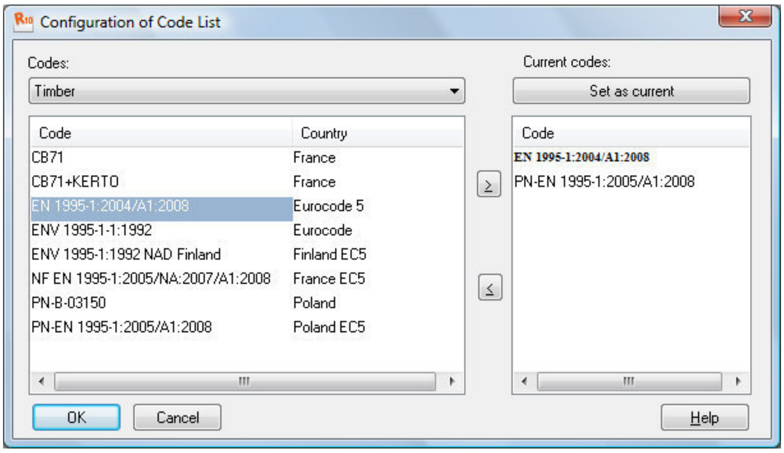
You can define a new type of Job Preferences to make it easier in the future.
First of all, make selection of documents and parameters appropriate for the project conditions from the list view tabs in JOB PREFERENCES dialog.
For example, to choose code, click *Design codes* tab from the left list view; then select code from *Timber structures* selection list or press *More codes* button which opens *Configuration of Code List:*



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



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



After the job preferences decisions are set, you can save it under a new name by pressing *Save Job Preferences* icon in the JOB PREFERENCES dialog.

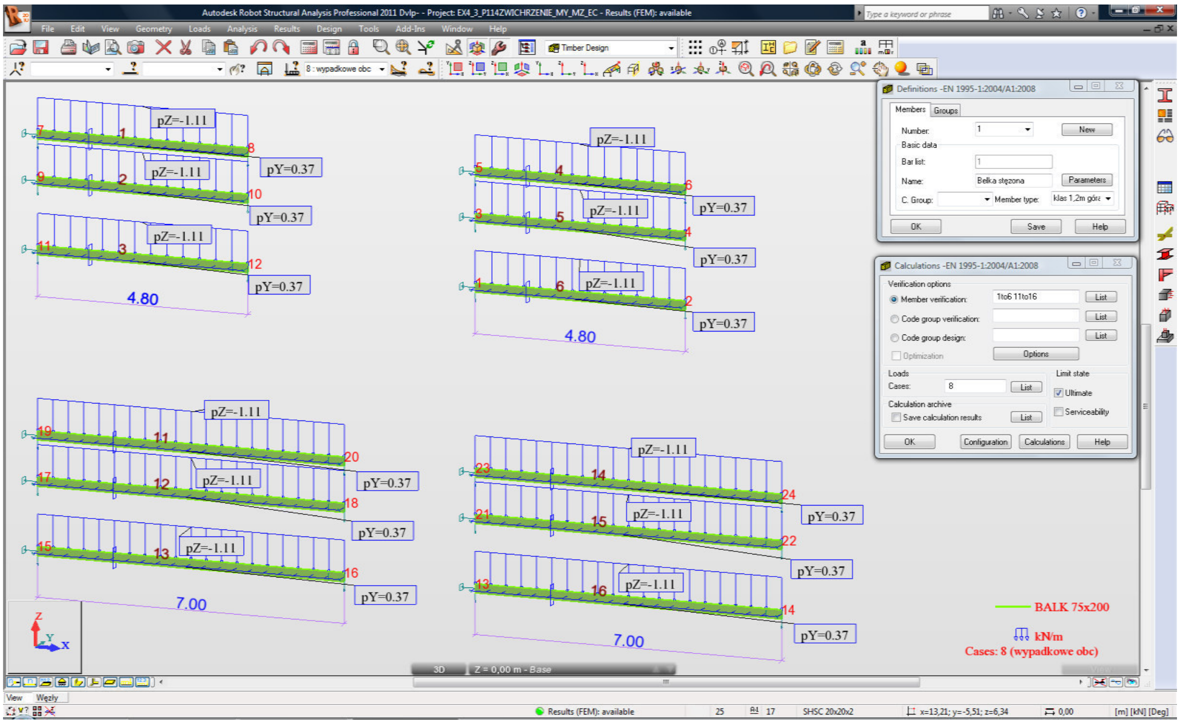
VERIFICATION PROBLEM 1
bending about two main axes with lateral buckling

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

TITLE:
Example 4.3 Solid Timber - Bending About Two Main Axes Restrained or Not Against Torsion
Eurocode5 - EN 1995-1-1:2004

SPECIFICATION:

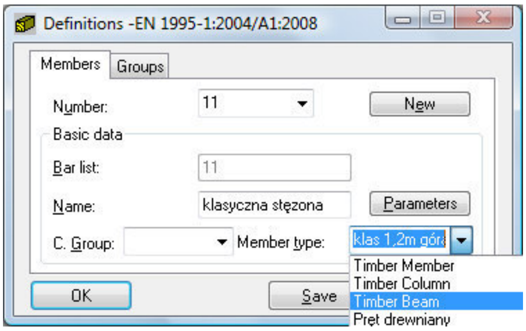
Verify the strength of the C16 cross-section 75×200 mm beams with simply supported spans of 4,8 m and 7,0 m. The beams n° 1, 4, 11, 14 are restrained at 1,2m against torsion. For load case n° 8 loads are assumed as a short-term load and are acting on the bottom (for el. n° 3, 6,13,16) or on the top of the beams (for the others elements) and are equal for all elements: py = 0,37 kN/m, pz = -1,11 kN/m.



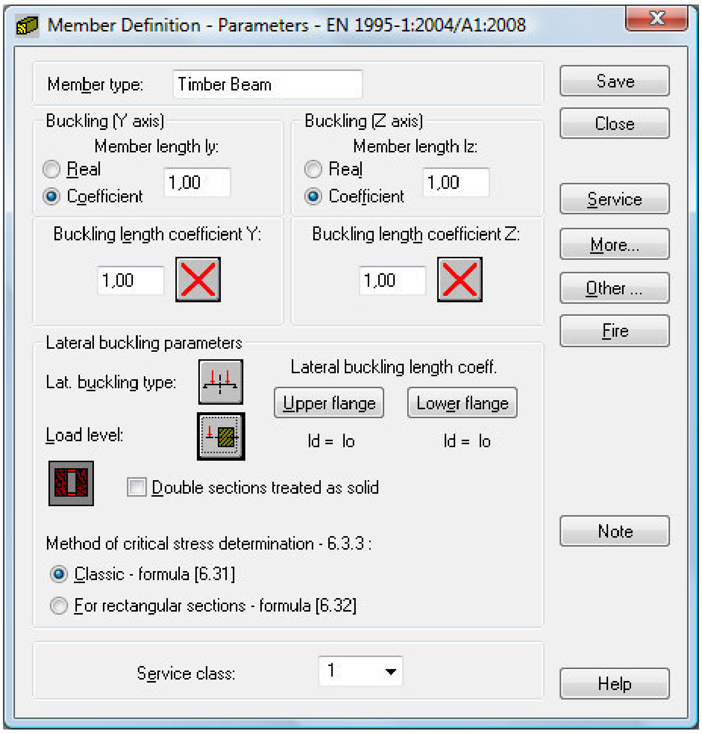
SOLUTION:

After having defined and calculated the structure models, go to [Timber Design] tab.
Define new types of members in accordance with the structure definition in DEFINITIONS dialog.
It can be set in *Member type* selection list.
In this example, the beams numbered 1, 4, 11, 14 are laterally braced at upper flange.

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

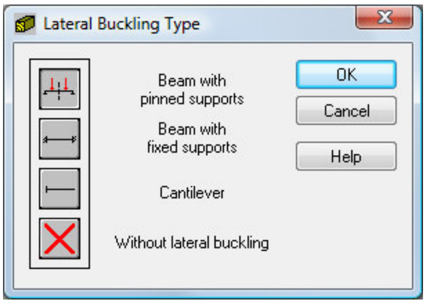


For the selected "Timber Beam" from member type, press the *Parameters* button on *Members* tab.
It opens MEMBER DEFINITION - PARAMETERS dialog.

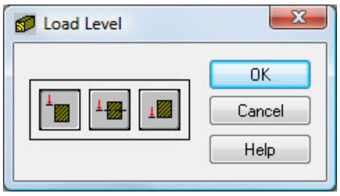


Type a new name in *Member type* editable field. Next, change the parameters to meet the initial data requirements of the structure. Set the following lateral-buckling parameters:

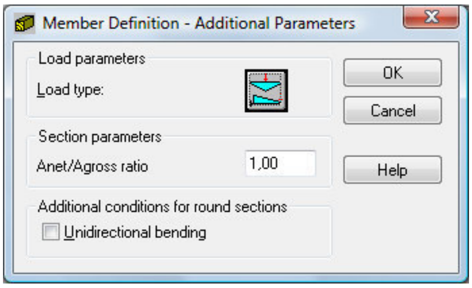
- switch on the appropriate *Lateral buckling type* icon*;*



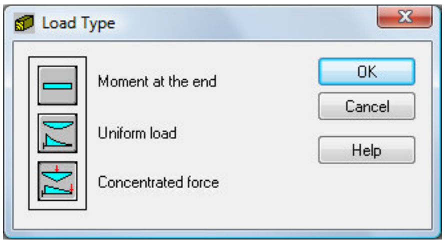
- select the appropriate *Load level* icon



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



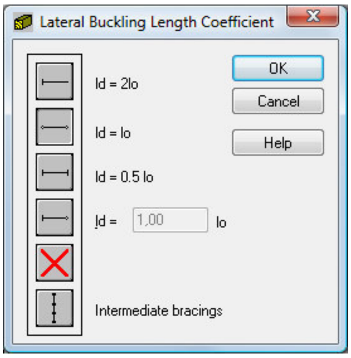
next, choose the load type by pressing the icon - it opens a new dialog:



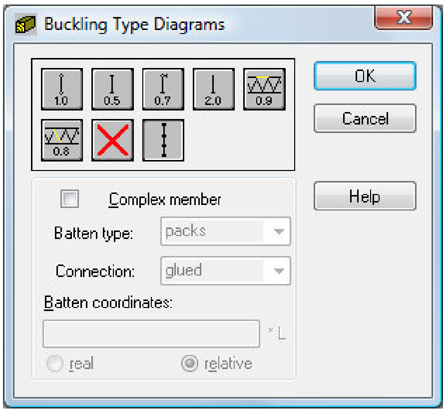
- select *Method of critical stress determination*
- choose *Service class*
- define bracings for *Lateral buckling* and B*uckling:*

→ to define *Lateral buckling length coefficient* for a member, press *Upper/Lower flange* button or the buckling type icon in [MEMBER DEFINITION-MEMBER] dialog

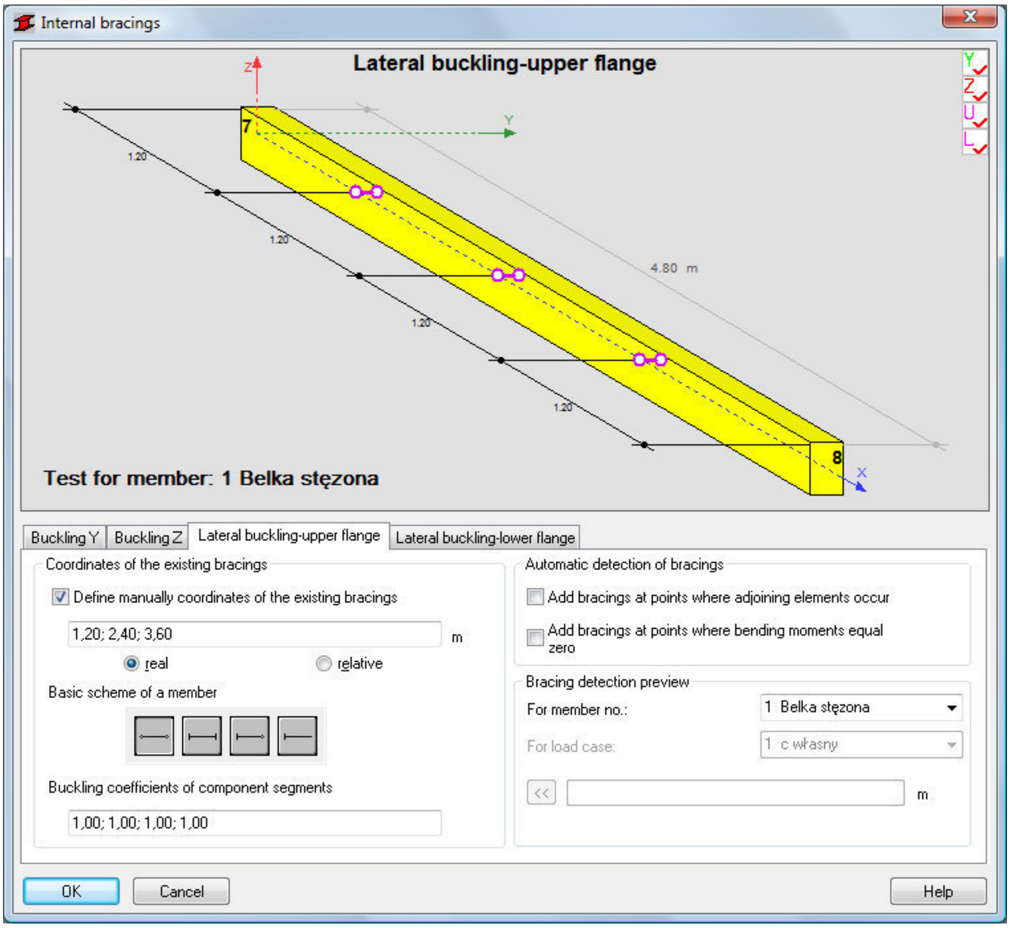
The first method opens LATERAL BUCKLING LENGTH COEFFICIENTS dialog:



The second one opens BUCKING TYPE DIAGRAMS dialog:

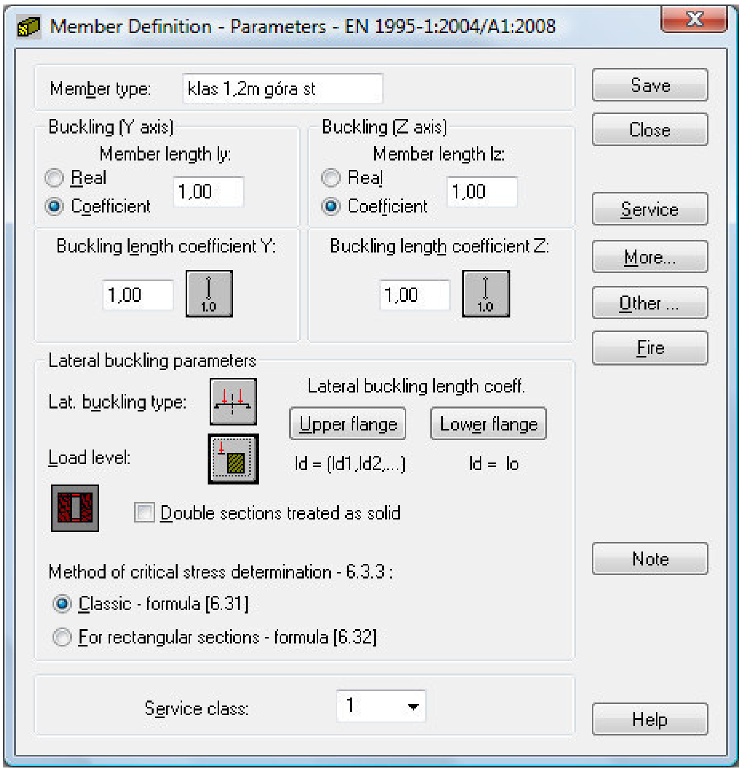


If you click the last icon - *Intermediate bracings* - the new dialog INTERNAL BRACINGS will appear.



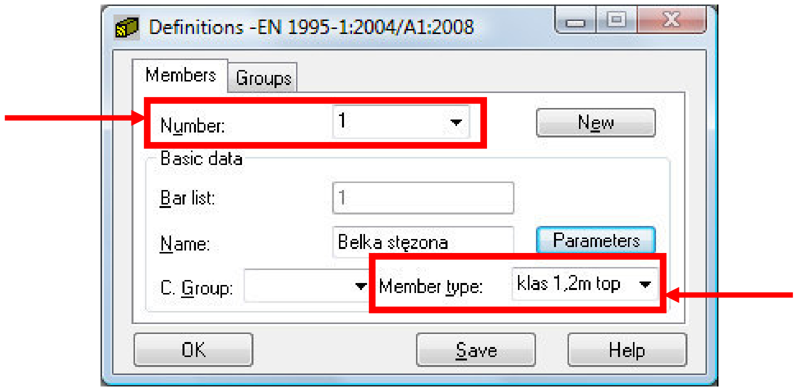
In the *INTERNAL BRACINGS* dialog, there are possibilities to define bracings for buckling and lateral buckling for the marked *member type* independently.
In this particular example of restrained elements, define member type with lateral buckling-upper flange internal bracings.

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

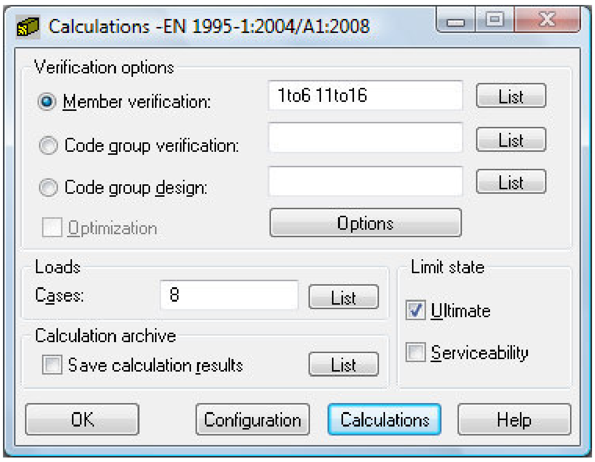


<u>Number</u> of the member must be assigned to the appropriate name of *<u>Member type</u>*.

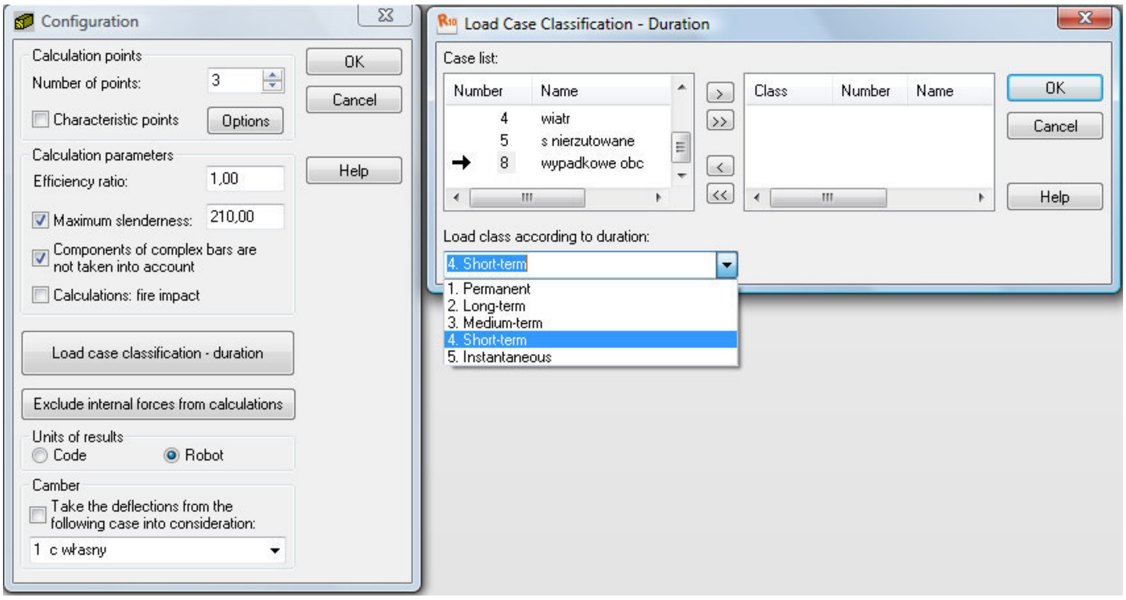
!!! It is very important when verifying different member types



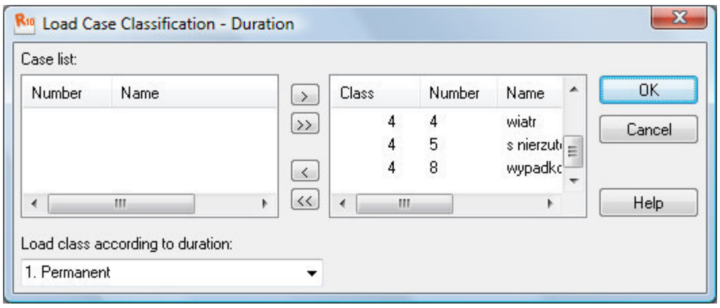
In the CALCULATIONS dialog set the following:
-> *Verification options - list of verified members*,
-> *Loads cases - list of chosen loads*
-> *Limit state*
->*Configuration*.



Before doing calculations you have to remember to specify appropriate duration for loads in the CALCULATIONS dialog:
- click [Configuration] button
- in CONFIGURATION dialog press [Load case classification - duration] button

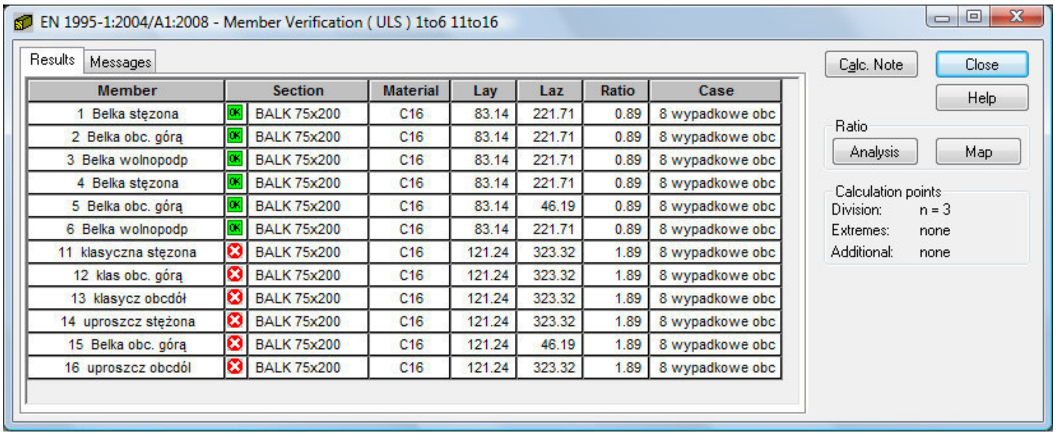


- in LOAD CASE CLASSIFICATION-DURATION dialog, assign "Load class according to duration" from selection list to the number of case list; for this particular example 4[th] "short-term" load case was selected and LOAD CASE CLASSIFICATION-DURATION dialog after the introduced changes looks as follows:



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

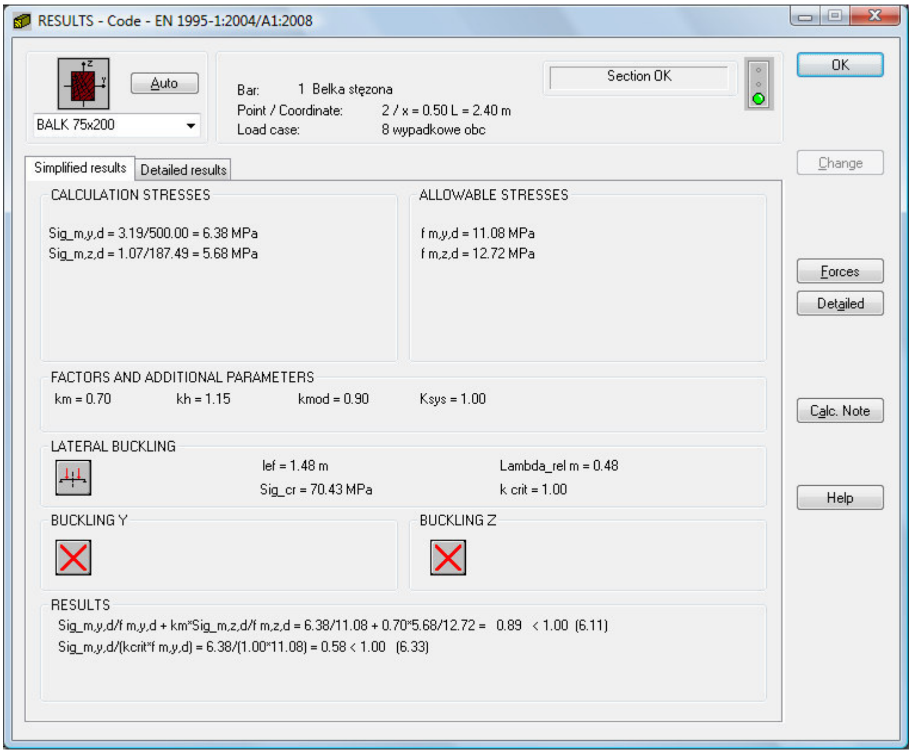
MEMBER VERIFICATION dialog with the most significant results data will appear on the screen.



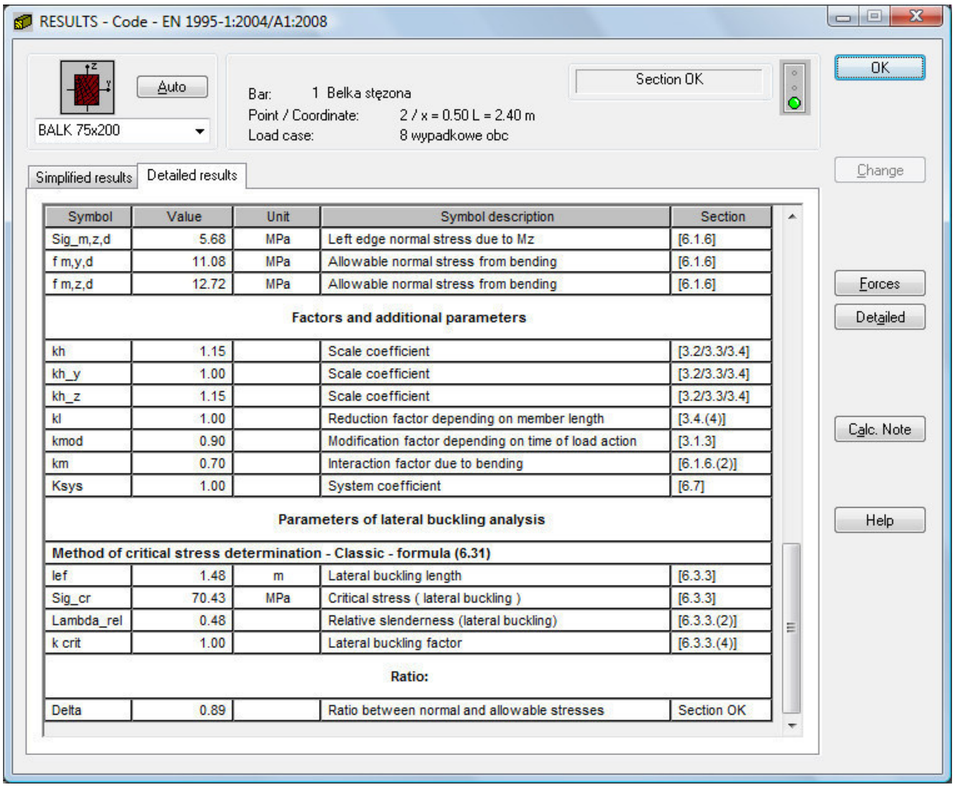
Member	Section	Material	Lay	Laz	Ratio	Case
1 Belka stężona	OK BALK 75x200	C16	83.14	221.71	0.89	8 wypadkowe obc
2 Belka obc. górą	OK BALK 75x200	C16	83.14	221.71	0.89	8 wypadkowe obc
3 Belka wolnopodp	OK BALK 75x200	C16	83.14	221.71	0.89	8 wypadkowe obc
4 Belka stęzona	OK BALK 75x200	C16	83.14	221.71	0.89	8 wypadkowe obc
5 Belka obc. górą	OK BALK 75x200	C16	83.14	46.19	0.89	8 wypadkowe obc
6 Belka wolnopodp	OK BALK 75x200	C16	83.14	221.71	0.89	8 wypadkowe obc
11 klasyczna stężona	BALK 75x200	C16	121.24	323.32	1.89	8 wypadkowe obc
12 klas obc. górą	BALK 75x200	C16	121.24	323.32	1.89	8 wypadkowe obc
13 klasycz obcdół	BALK 75x200	C16	121.24	323.32	1.89	8 wypadkowe obc
14 uproszcz stężona	BALK 75x200	C16	121.24	323.32	1.89	8 wypadkowe obc
15 Belka obc. górą	BALK 75x200	C16	121.24	46.19	1.89	8 wypadkowe obc
16 uproszcz obcdół	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



Detailed results tab



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 1995-1:2004/A1:2008*
ANALYSIS TYPE: Member Verification

CODE GROUP:
MEMBER: 1 Belka stężona **POINT:** 2 **COORDINATE:** x = 0.50 L = 2.40 m

---

LOADS:
Governing Load Case: 8 wypadkowe obc

MATERIAL C16

gM = 1.30	f m,0,k = 16.00 MPa	f t,0,k = 10.00 MPa	f c,0,k = 17.00 MPa
f v,k = 1.80 MPa	f t,90,k = 0.50 MPa	f c,90,k = 2.20 MPa	E 0,moyen = 8000.00 MPa
E 0,05 = 5400.00 MPa	G moyen = 500.00 MPa	Service class: 1	Beta c = 1.00

SECTION PARAMETERS: BALK 75x200

ht=20.0 cm			
bf=7.5 cm	Ay=40.91 cm2	Az=109.09 cm2	Ax=150.00 cm2
tw=3.8 cm	Iy=5000.00 cm4	Iz=703.10 cm4	Ix=2148.0 cm4
tf=3.8 cm	Wely=500.00 cm3	Welz=187.49 cm3	

STRESSES	**ALLOWABLE STRESSES**
Sig_m,y,d = MY/Wy= 3.19/500.00 = 6.38 MPa	f m,y,d = 11.08 MPa
Sig_m,z,d = MZ/Wz= 1.07/187.49 = 5.68 MPa	f m,z,d = 12.72 MPa

---

Factors and additional parameters
km = 0.70 kh = 1.15 kmod = 0.90 Ksys = 1.00

LATERAL BUCKLING PARAMETERS:

lef = 1.48 m	Lambda_rel m = 0.48
Sig_cr = 70.43 MPa	k crit = 1.00

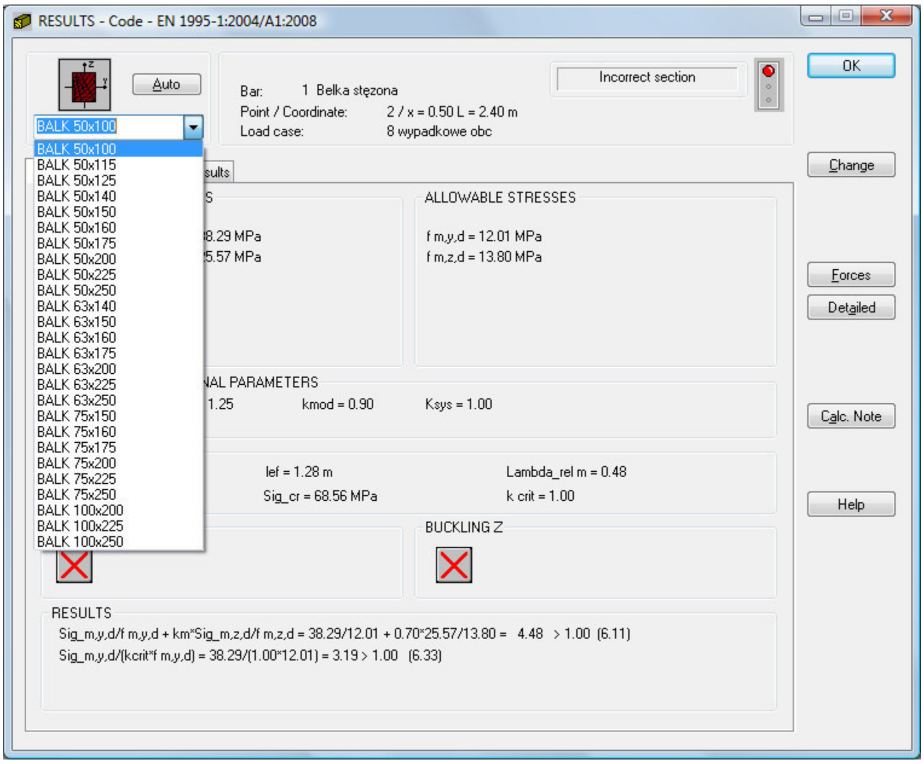
BUCKLING PARAMETERS:
About Y axis: About Z axis:

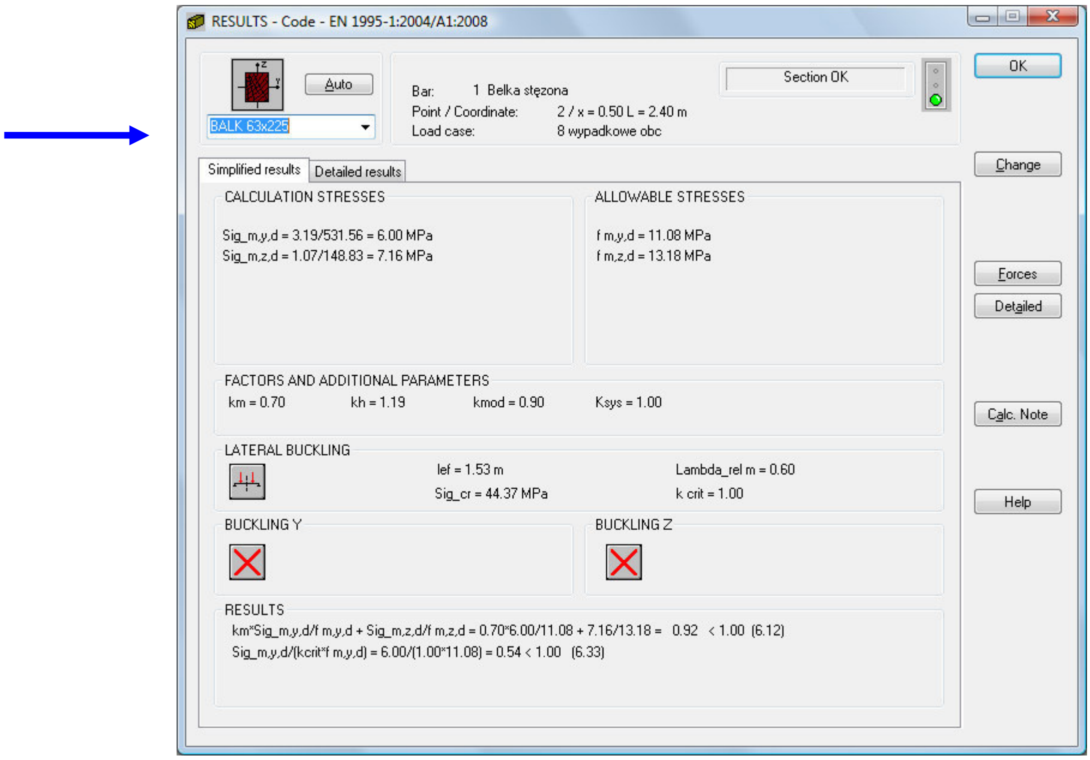
VERIFICATION FORMULAS:
Sig_m,y,d/f m,y,d + km*Sig_m,z,d/f m,z,d = 6.38/11.08 + 0.70*5.68/12.72 = 0.89 < 1.00 (6.11)
Sig_m,y,d/(kcrit*f m,y,d) = 6.38/(1.00*11.08) = 0.58 < 1.00 (6.33)

Section OK !!!

b) For economical reasons try to check the other, e.g. lighter BALK section.

While still in R*ESULTS- 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.





The results for the newly selected section are presented below.

TIMBER STRUCTURE CALCULATIONS for BALK 63x225

CODE: *EN 1995-1:2004/A1:2008*
ANALYSIS TYPE: Member Verification

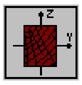
CODE GROUP:
MEMBER: 1 Belka stężona **POINT:** 2 **COORDINATE:** x = 0.50 L = 2.40 m

---

LOADS:
Governing Load Case: 8 wypa dkowe obc

MATERIAL C16

gM = 1.30	f m,0,k = 16.00 MPa	f t,0,k = 10.00 MPa	f c,0,k = 17.00 MPa
f v,k = 1.80 MPa	f t,90,k = 0.50 MPa	f c,90,k = 2.20 MPa	E 0,moyen = 8000.00 MPa
E 0,05 = 5400.00 MPa	G moyen = 500.00 MPa	Service class: 1	Beta c = 1.00



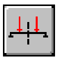
SECTION PARAMETERS: BALK 63x225

ht=22.5 cm			
bf=6.3 cm	Ay=31.02 cm2	Az=110.78 cm2	Ax=141.80 cm2
tw=3.1 cm	Iy=5980.10 cm4	Iz=468.80 cm4	Ix=1544.5 cm4
tf=3.1 cm	Wely=531.56 cm3	Welz=148.83 cm3	

STRESSES
Sig_m,y,d = MY/Wy= 3.19/531.56 = 6.00 MPa
Sig_m,z,d = MZ/Wz= 1.07/148.83 = 7.16 Mpa

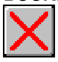
ALLOWABLE STRESSES
f m,y,d = 11.08 MPa
f m,z,d = 13.18 MPa

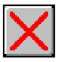
Factors and additional parameters
km = 0.70 kh = 1.19 kmod = 0.90 Ksys = 1.00



LATERAL BUCKLING PARAMETERS:
lef = 1.53 m Lambda_rel m = 0.60
Sig_cr = 44.37 MPa k crit = 1.00

BUCKLING PARAMETERS:


About Y axis:


About Z axis:

VERIFICATION FORMULAS:
km*Sig_m,y,d/f m,y,d + Sig_m,z,d/f m,z,d = 0.70*6.00/11.08 + 7.16/13.18 = 0.92 < 1.00 (6.12)
Sig_m,y,d/(kcrit*f m,y,d) = 6.00/(1.00*11.08) = 0.54 < 1.00 (6.33)

Section OK !!!

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

verification parameters, interaction expression		Robot	Handbook
L - beam length	[m]	4,8	4,8
Leff - effective length of the beam (Table 6.1, EC5)	[m]	1,48	1,48
σ m,cr = f (Leff) - critical bending stress	[MPa]	70,43	70,43
σ m,y,d - design bending stress due to My	[MPa]	6,382	6,39
σ m,z,d - design bending stress due to Mz	[MPa]	5,68	5,68
f m,y,d - design bending strength due to My	[MPa]	11,08	11,08
f m,z,d - design bending strength due to Mz	[MPa]	12,72	12,74
ratio (6.11) → σ m,y,d /f m,y,d + km* σ m,,z,d/f m,z,d =		0,889	0,89

CONCLUSIONS:

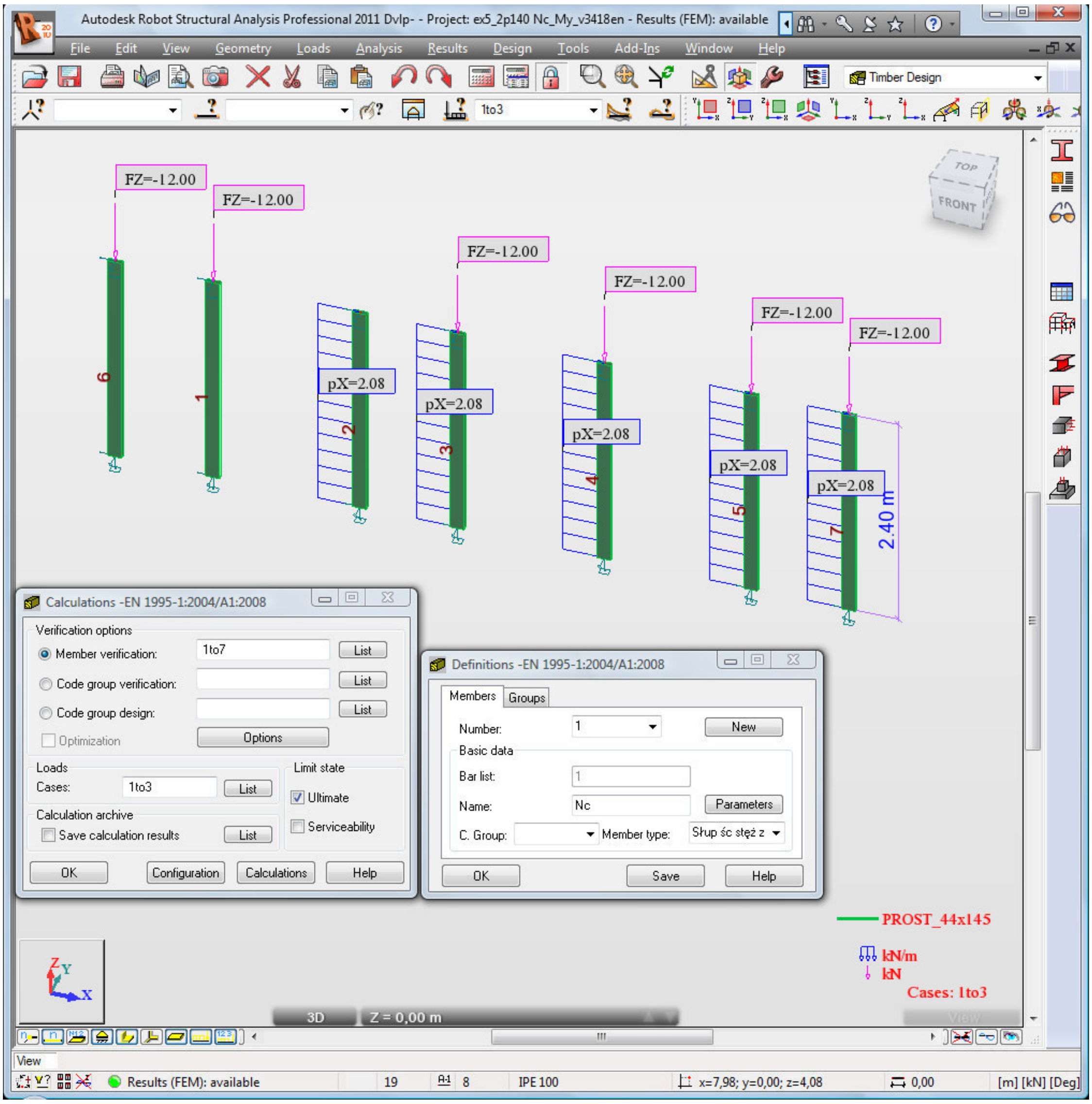
Agreement of results.
The small differences are caused by different accuracy of parameters in calculations.

VERIFICATION PROBLEM 2
combined compression and bending about one main axis

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

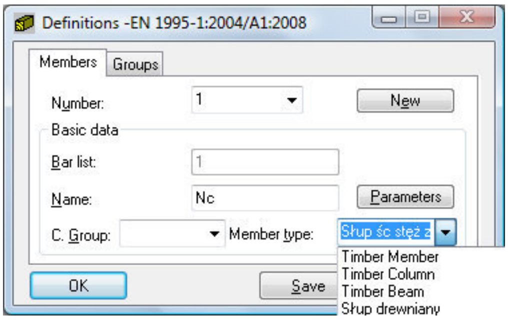
TITLE:
Example 5.2 - Solid Shape Subjected to Combined Compression and Bending About One Main Axis

SPECIFICATION:
Verify if a simply supported rectangular columns of C16 with planed cross-section 44x145mm have sufficient available strength to support a permanent concentric compression load *Fz* = 12 kN and uniformly distributed lateral wind load inducing a design moment My = 1,5 kNm at mid-span about the strong axis. The unbraced length is 2,4m and Service Class 2. There are different types of buckling parameters for columns.

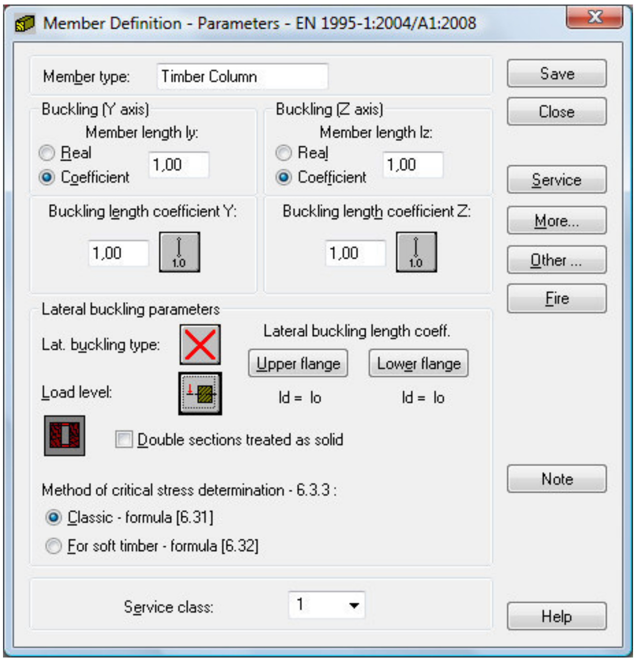


SOLUTION:

After having defined and calculated the structure model, go to [Timber Design] tab.
In DEFINITIONS dialog, define a new type of member. It can be set in *Member type* combo-box.
Pre-defined type of member, e.g. "*timber column*" may be initially opened.

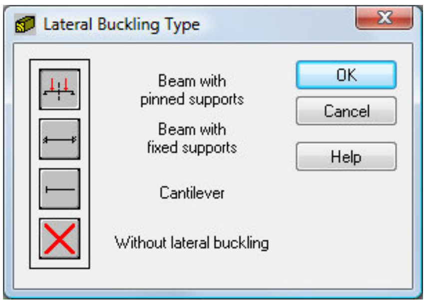


For the selected member type, press the *Parameters* button on *Members* tab.
The MEMBER DEFINITION-PARAMETERS dialog opens.

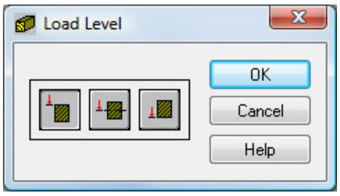


Type a new name in the *Member type* editable field. Next, change the parameters to meet the initial data requirements of a structure, e.g.:

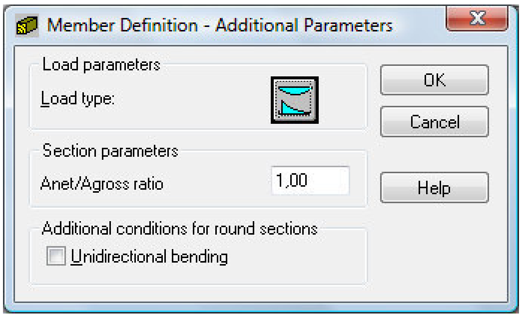
- switch on the appropriate *Lateral buckling type* icon;



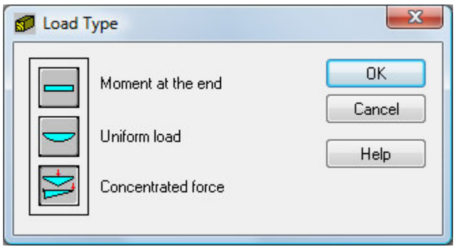
- select appropriate *Load level* icon



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



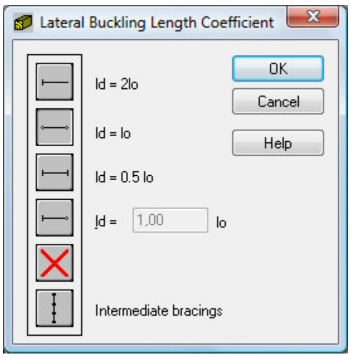
pressing the *Load type* icon opens a new dialog in which load type can be selected



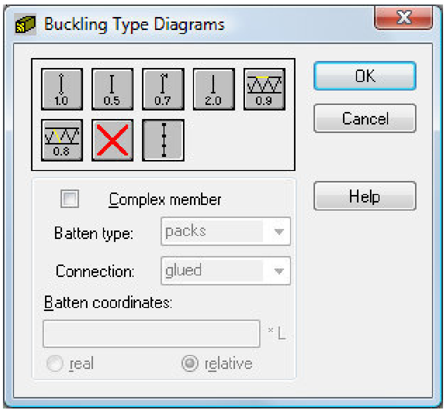
- define bracings for *Lateral buckling* and B*uckling.*

To define *Lateral buckling length coefficient* for a member, press *Upper/Lower flange* button or buckling type icon in [MEMBER DEFINITION-MEMBER] dialog.

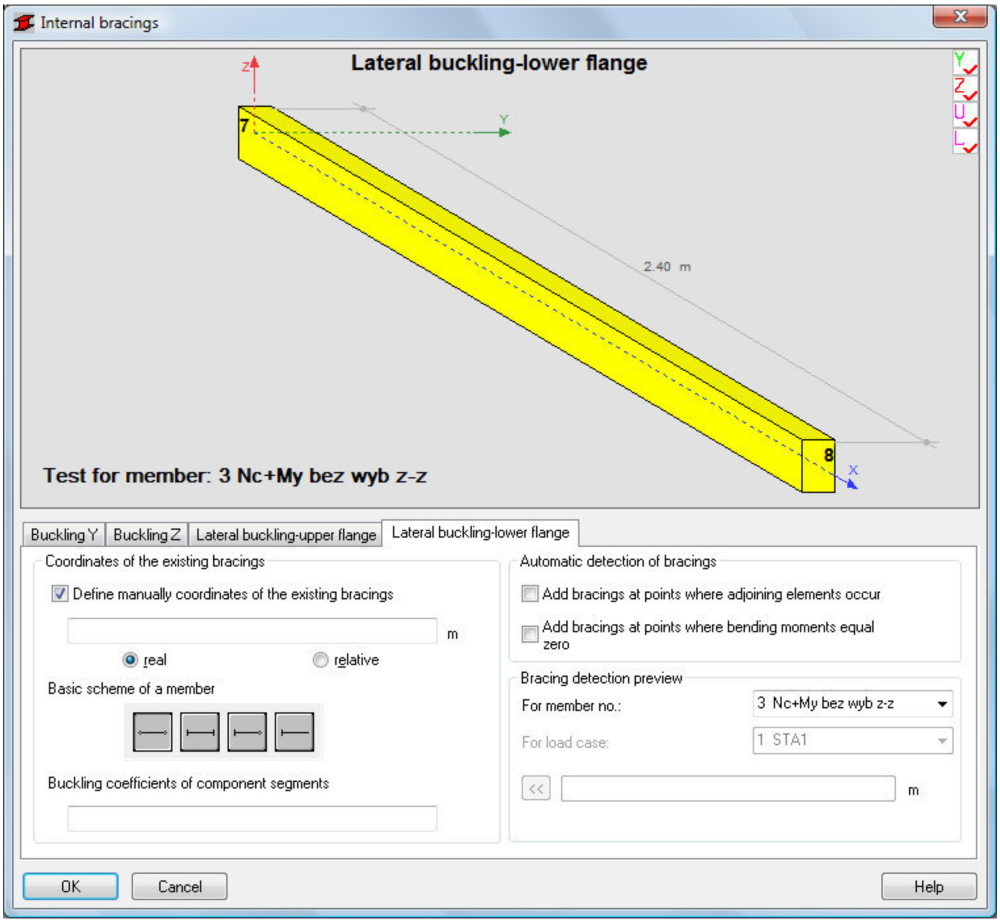
The first method opens LATERAL BUCKLING LENGTH COEFFICIENTS dialog,



the second opens > BUCKING TYPE DIAGRAMS dialog.

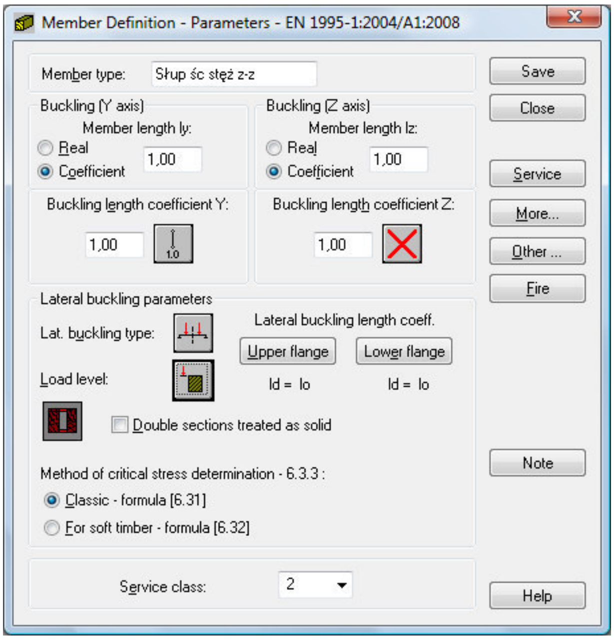


If you click the last icon *Intermediate bracings,* the new dialog INTERNAL BRACINGS will appear:



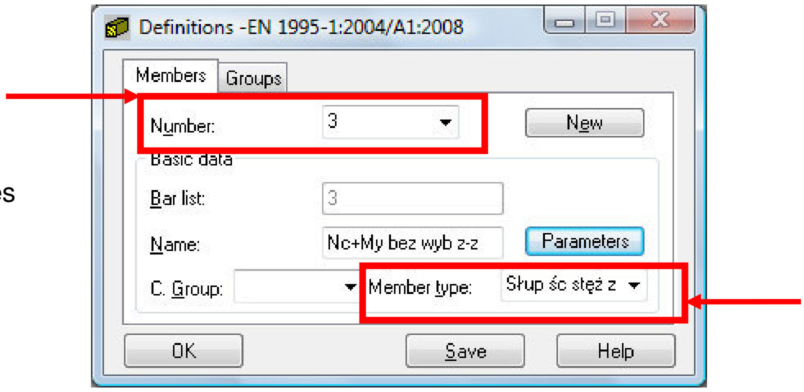
There are possibilities to define independently bracings for buckling and lateral buckling for the marked *member type* in *INTERNAL BRACINGS* dialog.

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:

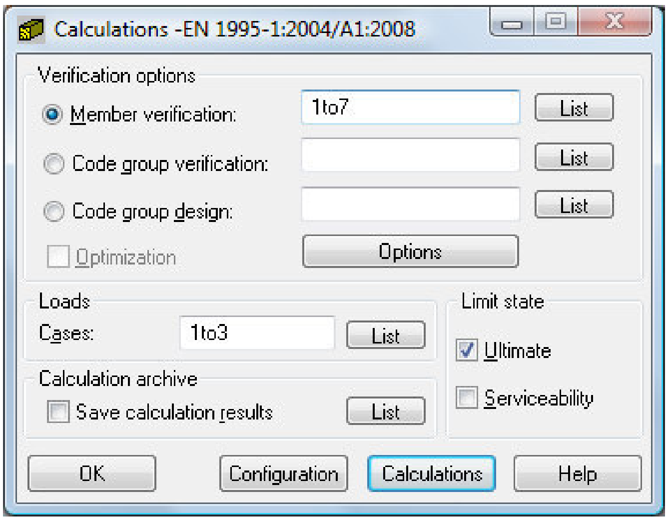


The *Number* of the member must be assigned to appropriate name of *Member type*

→ it is very important when verifying different member types



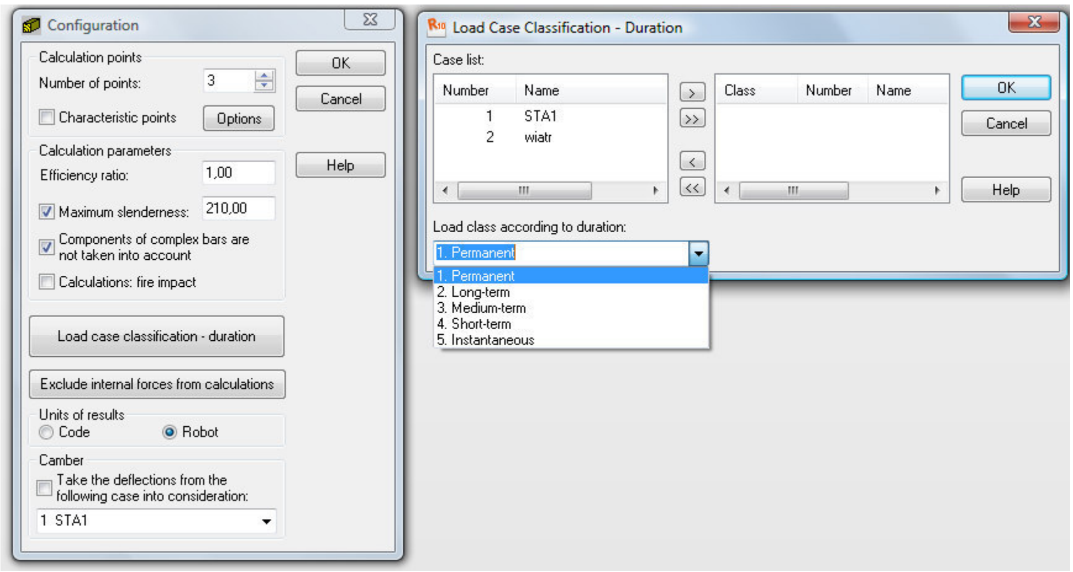
In CALCULATIONS dialog, set the following:
-> *Verification options - list of verified members*
-> *Loads cases - list of chosen loads*
-> *Limit state*
-> *Configuration.*



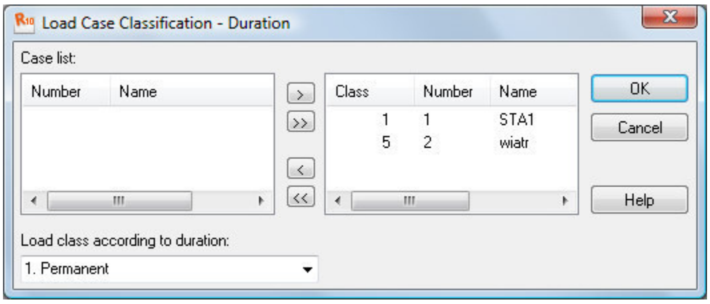
Before you verify the member, you have to specify appropriate duration for loads in CALCULATIONS dialog:

- click [Configuration] button

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

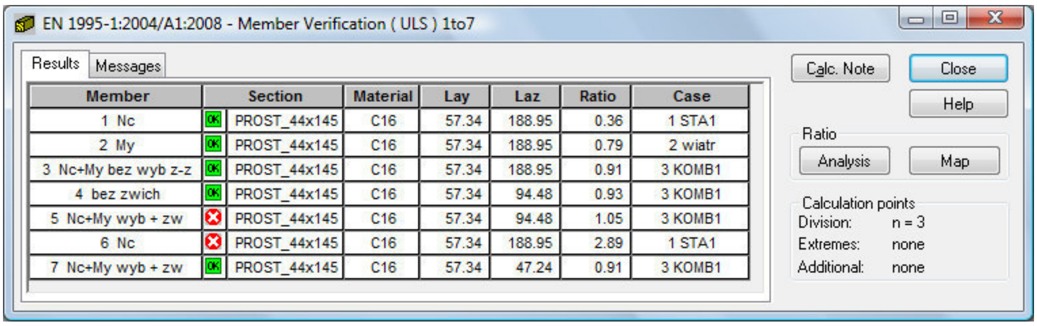


- in LOAD CASE CLASSIFICATION-DURATION dialog, assign “Load class according to duration” from combo box list to the number of the case list; in this particular example, the first “permanent” and the fifth “instantaneous” load case were selected and LOAD CASE CLASSIFICATION-DURATION dialog after the introduced changes looks as follows:



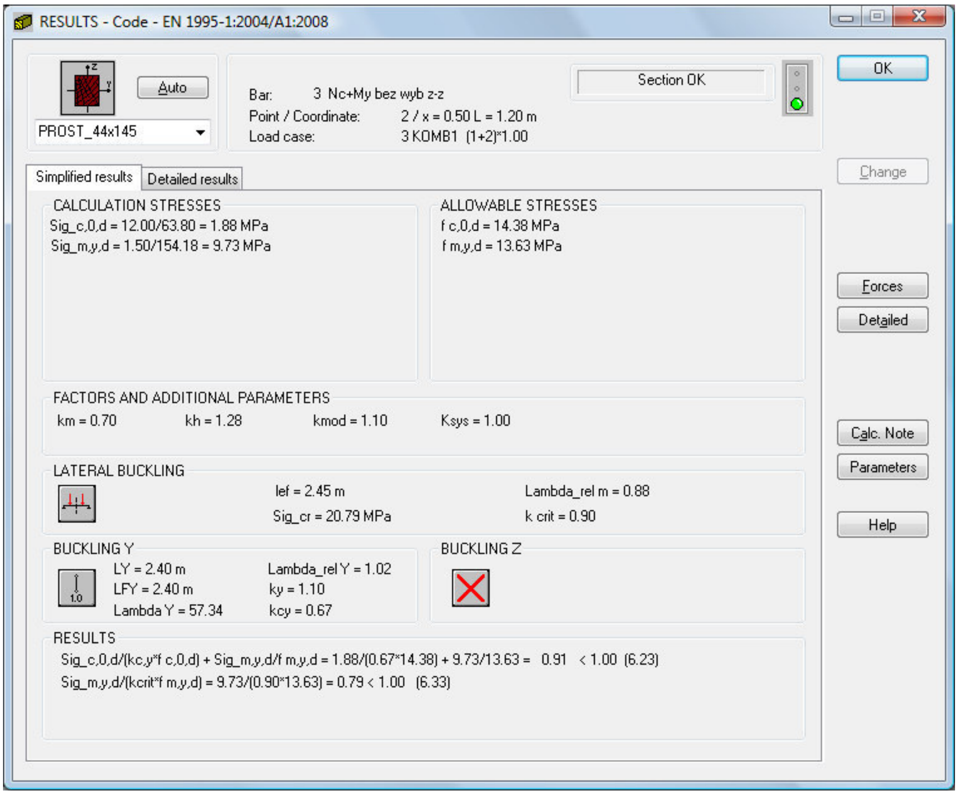
Start verification by pressing *Calculations* button in CALCULATIONS dialog.

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

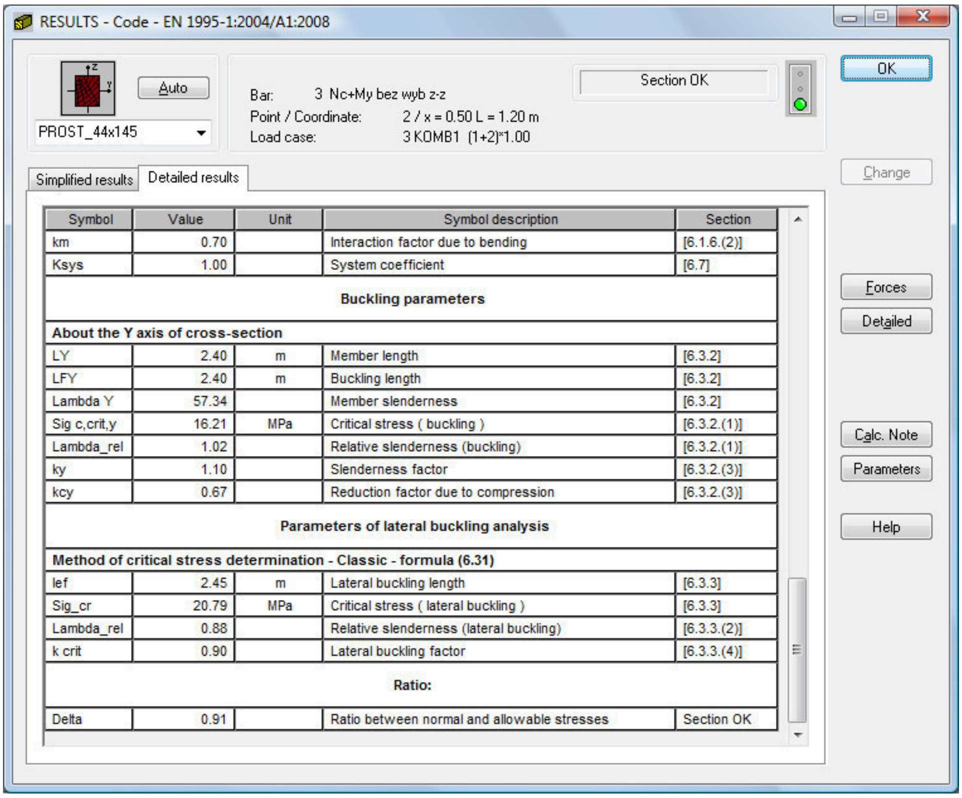


Pressing the line with the result for any member opens the RESULTS dialog with more detailed results for the analyzed member. The views of the RESULTS dialogs, e.g. for the third member, are presented below.

Simplified results tab



Detailed results tab



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 1995-1:2004/A1:2008*
ANALYSIS TYPE: Member Verification

CODE GROUP:
MEMBER: 3 Nc+My bez wyb z-z **POINT:** 2 **COORDINATE:** x = 0.50 L = 1.20 m

---

LOADS:
Governing Load Case: 3 KOMB1 (1+2)*1.00

MATERIAL C16

gM = 1.30	f m,0,k = 16.00 MPa	f t,0,k = 10.00 MPa	f c,0,k = 17.00 MPa
f v,k = 1.80 MPa	f t,90,k = 0.50 MPa	f c,90,k = 2.20 MPa	E 0,moyen = 8000.00 MPa
E 0,05 = 5400.00 MPa	G moyen = 500.00 MPa	Service class: 2	Beta c = 0.20

SECTION PARAMETERS: PROST_44x145

ht=14.5 cm

bf=4.4 cm	Ay=14.85 cm2	Az=48.95 cm2	Ax=63.80 cm2
tw=2.2 cm	Iy=1117.83 cm4	Iz=102.93 cm4	Ix=333.0 cm4
tf=2.2 cm	Wely=154.18 cm3	Welz=46.79 cm3	

STRESSES	**ALLOWABLE STRESSES**
Sig_c,0,d = N/Ax = 12.00/63.80 = 1.88 MPa	f c,0,d = 14.38 MPa
Sig_m,y,d = MY/Wy= 1.50/154.18 = 9.73 MPa	f m,y,d = 13.63 MPa

---

Factors and additional parameters
km = 0.70 kh = 1.28 kmod = 1.10 Ksys = 1.00

LATERAL BUCKLING PARAMETERS:

lef = 2.45 m	Lambda_rel m = 0.88
Sig_cr = 20.79 MPa	k crit = 0.90

BUCKLING PARAMETERS:

About Y axis: About Z axis:

LY = 2.40 m	Lambda Y = 57.34
Lambda_rel Y = 1.02	ky = 1.10
LFY = 2.40 m	kcy = 0.67

VERIFICATION FORMULAS:
Sig_c,0,d/(kc,y*f c,0,d) + Sig_m,y,d/f m,y,d = 1.88/(0.67*14.38) + 9.73/13.63 = 0.91 < 1.00 (6.23)
Sig_m,y,d/(kcrit*f m,y,d) = 9.73/(0.90*13.63) = 0.79 < 1.00 (6.33)

Section OK !!!

COMPARISON:

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

verifications parameters, interaction expression	Robot	Handbook
λy - member slenderness	57,34	57,3
ky - slenderness factor	1,097	1,097
kcy - reduction factor due to compression	0,671	0,671
k_{mod}	1,1	1,1
f c,o,d - design compression strength [MPa]	14,38	14,38
f m,y,d - design bending strength due to My [MPa]	13,63	13,54
σ c,o,d - design compression stress [MPa]	1,88	1,88
σ m,y,d - design bending stress due to My [MPa]	9,73	9,73
ratio from (6.23) → σ c,o,d / ($k_{c,y}$ *fc,o,d) + σ m,y,d / f m,y,d =	**0,91**	**0,91**

CONCLUSIONS:

Total agreement of results.

Autodesk®
Robot™ Structural Analysis Professional

VERIFICATION MANUAL

March 2014

according to:

NAFEMS benchmarks:

- Linear Static Benchmarks vol. 1. (Ref: LSB 1)
- Selected Benchmarks for Forced Vibration (Ref: R0016)
- Background to FE Analysis of Geometric Non-linearity Benchmarks (Ref: R0065)

Table of contents

INTRODUCTION

This report contains a range of static and dynamic benchmark tests covering a few types of behaviour encountered in structural analysis.
These examples have been taken from:

"Linear Static Benchmarks vol.1" signed by NAFEMS as LSB1,
“Selected Benchmarks for Forced Vibration” signed by NAFEMS as R0016;
“Background to FE Analysis of Geometric Non-linearity Benchmarks” signed by NAFEMS as R0065.

Benchmark results (signed as “NAFEMS”) were recalled, and originally compared with results of **Autodesk Robot Structural Analysis Professional version 2013** (signed further as “Robot”). The comparison of results is still valid for the next versions.

Each problem contains the following parts:

- the name of the benchmark as used in NAFEMS manual,
- short problem description,
- scheme of the model,
- comparison between Robot results and reference values.

SHELL

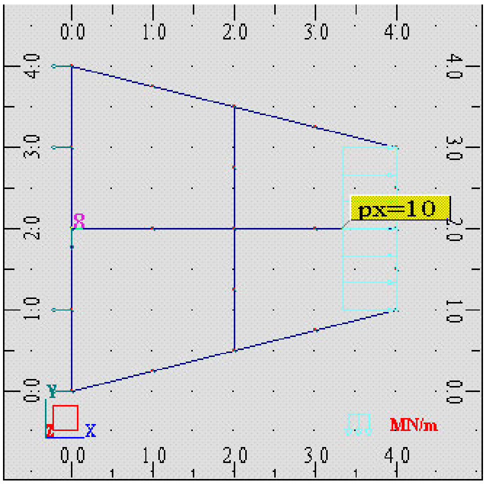
LINEAR STATIC ANALYSIS

VERIFICATION EXAMPLE

TEST IC1 - tapered membrane end load

Name of the test: IC1
Reference: NAFEMS LSB1
Specification: Linear static analysis of an elastic membrane.

GEOMETRY: Thickness = 0.1 m



DATA DEFINITION:

Loading: Uniformly distributed horizontal load of 10 MN/m (pressure of 100 Mpa) along outer edge.
Boundary conditions: Nodes on X=0.0 – blocked UX, UZ, RY, node (0.0, 2.0) - fully clamped.
Material properties: Isotropic, E=210e3 MPa, ni=0.3
Element type: Shell 8-node quadrilaterals
DATA FILE: Nafems_IC01.rtd

RESULTS COMPARISON:

	Direct stress s_{xx} at point no. 8 (0.0, 2.0)		
Mesh refinement	**NAFEMS**	**Robot**	**Difference**
2x2	61.9	65.38	6.66%
4x4	60.9	61.79	0.80%
8x8	61.3	61.53	0.38%
16x16	---	61.39	0.15%
32x32	---	61.35	0.08%

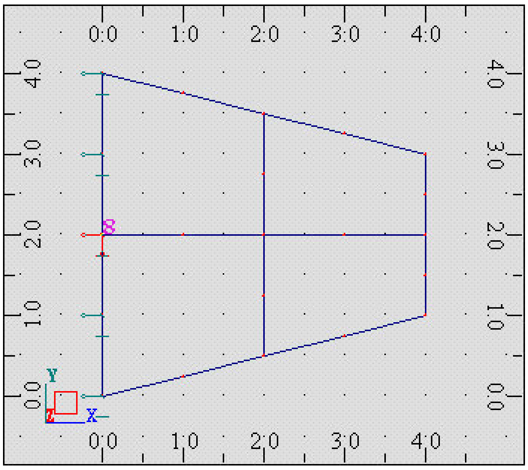
TARGET: 61.3 MPa

VERIFICATION EXAMPLE

TEST IC2 - tapered membrane gravity loading

Name of the test: IC2
Reference: NAFEMS LSB1
Specification: Linear static analysis of an elastic membrane.

GEOMETRY: Thickness = 0.1 m



DATA DEFINITION:

Loading: Uniform acceleration 9.81 m/s2 in global X direction (gravity).
Boundary conditions: Nodes on X=0.0 – blocked UX, UZ, RY, node (0.0, 2.0) - fully clamped.
Material properties: Isotropic, E=210e3 MPa, ni=0.3, p=7 MG/m3
Element type: Shell 8-node quadrilaterals
DATA FILE: Nafems_IC02.rtd

RESULTS COMPARISON:

	Direct stress s_{xx} at point no. 8 (0.0, 2.0)		
Mesh refinement	**NAFEMS**	**Robot**	**Difference**
2x2	0.258	0.2697	9.19%
4x4	0.247	0.2487	0.69%
8x8	0.247	0.2477	0.28%
16x16	---	0.2471	0.04%
32x32	---	0.2469	0.04%

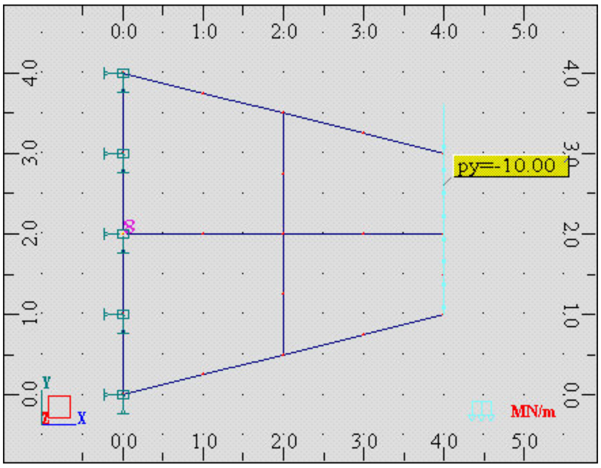
TARGET: 0.247 MPa

VERIFICATION EXAMPLE

TEST IC3 - tapered membrane edge shear

Name of the test: IC3
Reference: NAFEMS LSB1
Specification: Linear static analysis of an elastic membrane.

GEOMETRY: Thickness = 0.1 m



DATA DEFINITION:

Loading: Uniform surface shear traction of 100 Mpa in the vertical Y- direction.
Boundary condition: Edge X=0.0 – fully fixed.
Material properties: Isotropic, E=210e3 MPa, ni=0.3
Element type: Shell 8-node quadrilaterals
DATA FILE: Nafems_IC03.rtd

RESULTS COMPARISON:

	Direct stress s_{xy} at point no. 8 (0.0, 2.0)		
Mesh refinement	**NAFEMS**	**Robot**	**Difference**
2x2	28.7	26.889	0.04%
4x4	27.9	27.166	0.99%
8x8	27.3	27.157	0.96%
16x16	---	26.945	0.17%
32x32	---	26.862	0.14%

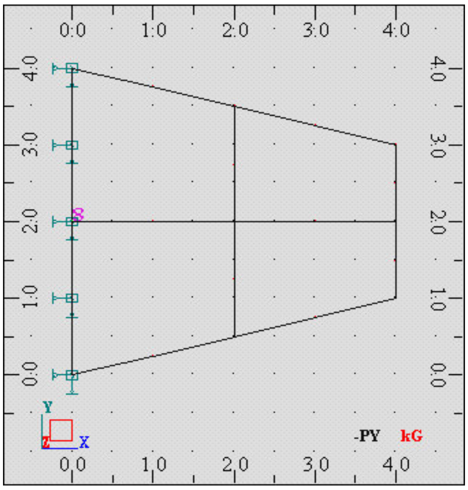
TARGET: 26.9 MPa

VERIFICATION EXAMPLE

TEST IC4 - tapered cantilever gravity load

Name of the test: IC4
Reference: NAFEMS LSB1
Specification: Linear static analysis of an elastic plate

GEOMETRY: Thickness = 0.1 m



DATA DEFINITION:

Loading: Uniform acceleration 9.81 m/s2 in the vertical Y direction (gravity).
Boundary condition: Edge X=0.0 – fully fixed.
Material properties: Isotropic, E=210e3 MPa, ni=0.3, p=7 MG/m3
Element type: shell 8-node quadrilaterals
DATA FILE: Nafems_IC04.rtd

RESULTS COMPARISON:

	Direct stress s_{xy} at point no. 8 (0.0, 2.0)		
Mesh refinement	**NAFEMS**	**Robot**	**Difference**
2x2	0.198	0.1831	8.45%
4x4	0.200	0.2000	0.00%
8x8	0.200	0.1992	0.40%
16x16	---	0.1990	0.50%
32x32	---	0.1991	0.45%

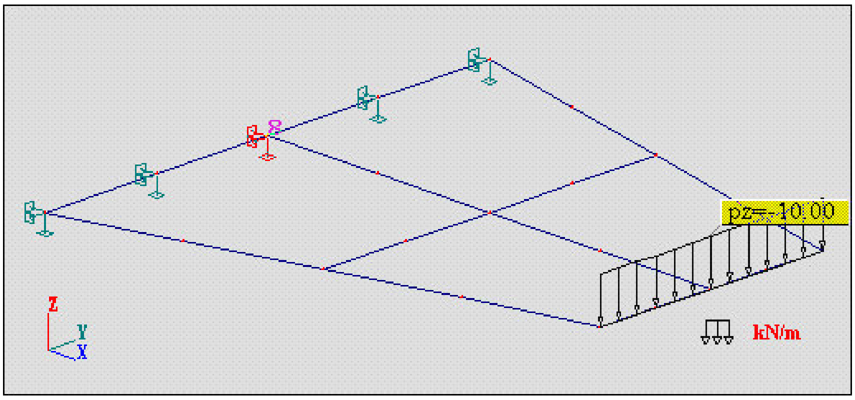
TARGET: 0.2000 MPa

VERIFICATION EXAMPLE

TEST IC10 - tapered plate edge shear

Name of the test: IC10
Reference: NAFEMS LSB1
Specification: Linear static analysis of an elastic plate

GEOMETRY: Thickness = 0.1 m



DATA DEFINITION:

Loading: Uniform vertical shear 10 kN/m in the Z direction along outer edge.
Boundary condition: Edge X=0.0 – fully fixed.
Material properties: Isotropic, E=210e3 MPa, ni=0.3
Element type: shell 8-node quadrilaterals
DATA FILE: Nafems_IC10.rtd

RESULTS COMPARISON:

	Value of Direct stress s_{xx} on top surface at point no 8 (0.0, 2.0)		
Mesh refinement	**NAFEMS**	**Robot**	**Difference**
2x2	14.3	14.750	0.34%
4x4	14.5	14.684	0.11%
8x8	14.6	14.610	0.61%
16x16	---	14.626	0.50%
32x32	---	14.630	0.48%

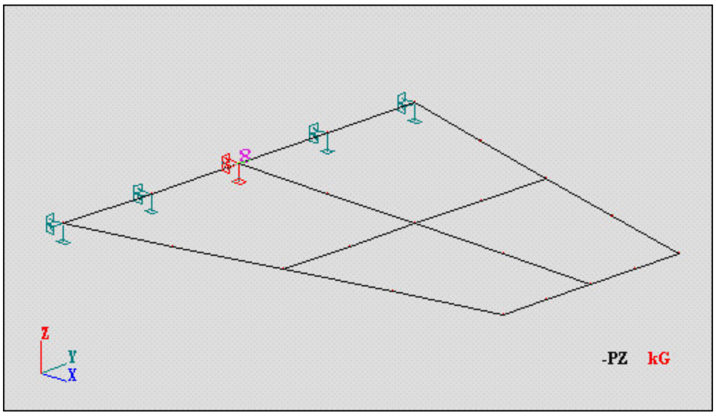
TARGET: 14.7 MPa

VERIFICATION EXAMPLE

TEST IC11 - tapered plate gravity load

Name of the test: IC11
Reference: NAFEMS LSB1
Specification: Linear static analysis of an elastic plate

GEOMETRY: Thickness = 0.1 m



DATA DEFINITION:

Loading: Uniform acceleration 9.81 m/s2 in the vertical Z direction (gravity).
Boundary condition: Edge X=0.0 – fully fixed.
Material properties: Isotropic, E=210e3 MPa, ni=0.3
Element type: shell 8-node quadrilaterals
DATA FILE: Nafems_IC11.rtd

RESULTS COMPARISON:

	Value of Direct stress s_{xx} on top surface at point no 8 (0.0, 2.0)		
Mesh refinement	**NAFEMS**	**Robot**	**Difference**
2x2	24.0	24.506	5.75%
4x4	25.5	25.675	1.25%
8x8	25.7	25.748	0.97%
16x16	---	25.830	0.65%
32x32	---	25.851	0.57%

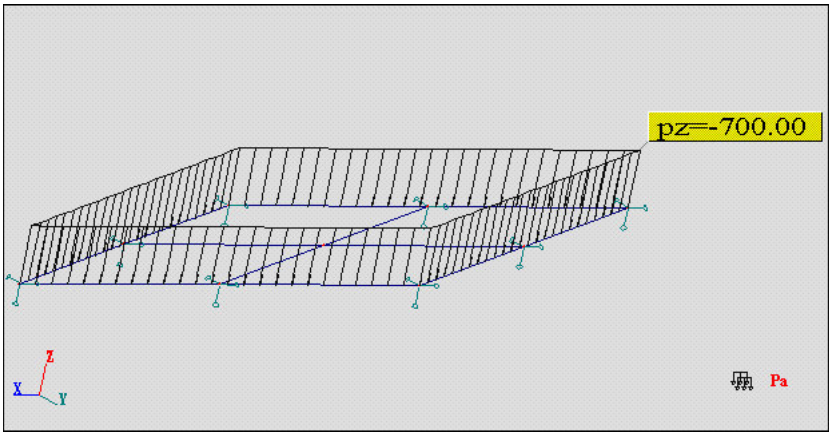
TARGET: 26 MPa

VERIFICATION EXAMPLE

TEST IC13 - skew plate normal pressure

Name of the test: IC13
Reference: NAFEMS LSB1
Specification: Linear static analysis of an elastic plate

GEOMETRY: Thickness = 0.01 m



DATA DEFINITION:

Loading: Normal pressure –0.7 kPa in the vertical Z direction.
Boundary condition: Simple supports (no z-displacement) for all edges.
Material properties: Isotropic, E=210e3 MPa, ni=0.3
Element type: shell 8-node quadrilaterals
DATA FILE: Nafems_IC13.rtd

RESULTS COMPARISON:

	Maximum principal stress on the lower surface at the plate center		
Mesh refinement	**NAFEMS**	**Robot**	**Difference**
2x2	0.757	0.5253	34.50 %
4x4	0.795	0.7214	10.04 %
8x8	---	0.7163	10.68 %
16x16	---	0.7678	4.26 %
32x32	---	0.7998	0.27 %

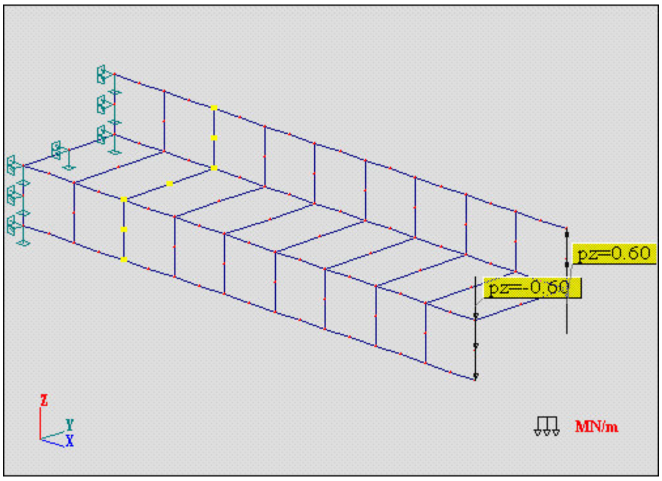
TARGET: 0.802 MPa

VERIFICATION EXAMPLE

TEST IC29 - Z-section cantilever torsion bending

Name of the test: IC29
Reference: NAFEMS LSB1
Specification: Linear static analysis of an elastic shell

GEOMETRY: Thickness = 0.1 m



DATA DEFINITION:

Loading:	Torque of 1.2 MNm at end x=10, by two uniformly distributes edge shears, S=0.6 MN at each flange.
Boundary condition:	At edge x=0 all displacements are zero.
Material properties:	Isotropic, E=210e3 MPa, ni=0.3
Element type:	Shell 8-node quadrilaterals
DATA FILE:	Nafems_IC29.rtd

RESULTS COMPARISON:

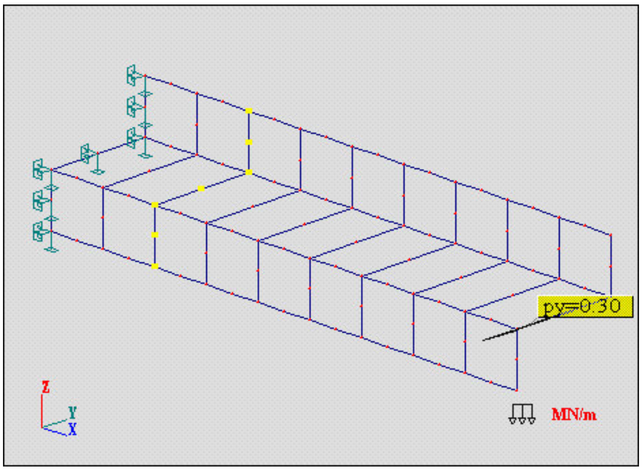
	Direct stress s_{xx} at mid surface at points in ¼ of the beam's length			
Point	**Target**	**NAFEMS**	**Robot**	**Difference**
1	-108.8	-110.1	-110.10	1.19%
2	-36.26	-36.9	-36.88	1.71%
3	36.26	36.2	36.04	0.61%
4	36.26	37.3	37.72	4.03%
5	36.26	36.2	36.04	0.61%
6	-36.26	-36.9	-36.88	1.71%
7	-108.8	-110.1	-110.10	1.19%

VERIFICATION EXAMPLE

TEST IC30 - Z-section cantilever beam bending

Name of the test: IC30
Reference: NAFEMS LSB1
Specification: Linear static analysis of an elastic shell

GEOMETRY: Thickness = 0.1 m



DATA DEFINITION:

Loading: Shear force S=0.6 MN as a uniformly distributed edge shear on the central web.
Boundary condition: At edge x=0 all x-displacements are zero.
y-displacements are zero at the origin,
z-displacements are zero at the corners of the two flanges at (0,-1,-1) and (0,1,1)
Material properties: Isotropic, E=210e3 MPa, ni=0.3
Element type: Shell 8-node quadrilaterals
DATA FILE: Nafems_IC30.rtd

RESULTS COMPARISON:

	Direct stress s_{xx} at mid surface at points in ¼ of the beam's length			
Point	**Target**	**NAFEMS**	**Robot**	**Difference**
1	193.0	191.0	194.48	0.77%
2	-96.5	-96.7	-97.25	0.78%
3	-386.0	-383.0	-386.22	0.06%
4	0.0	0.0	0.0	0.00%
5	386.0	383.0	386.22	0.06%
6	96.5	96.7	97.25	0.78%
7	-193.0	-191.0	-194.48	0.77%

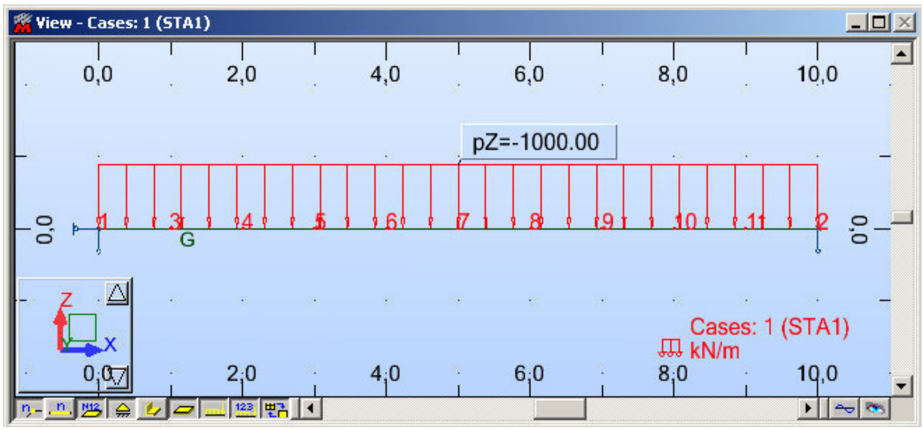
DYNAMIC ANALYSIS

VERIFICATION EXAMPLE

TESTS No 5 - Vibrations of a Deep Beam

Name of the test: 5, 5H, 5P, 5T
Reference: NAFEMS R0016
Specification: Dynamic analysis of an elastic beam

GEOMETRY: Length: L = 10 m
Section: a = b = 2 m



DATA DEFINITION:

Loading: Uniform load Fo=10^6 [N/m] (Fo=1000 [kN/m])
Boundary condition: X=Y=Z=RX=0 (at the beginning of the beam)
Y=Z=0 (at the end of the beam)
Material properties: E=$200x10^9$ N/m^2 ; ν=0,3 ; ρ=8000 kg/m^3
Element type: 10 beam elements, using attribute: 'Consider shear forces in deformation calculation' (Timoshenko's Beam – deep beam).
DATA FILE: Nafems_05.rtd

RESULTS COMPARISON:

Results of Modal Analysis (5)

OUTPUT: Frequencies [Hz]

Modes	NAFEMS	Robot	**Difference**
1&2	42.65	42.49	0.38%
3	71.20	71.26	0.08%
4	125.00	125.11	0.09%
5&6	148.15	143.79	3.15%
7	213.61	215.54	0.90%
8&9	283.47	259.36	8.51%

Results of Harmonic Forced Vibration (5H)

Forcing function: F=Fosin(2πft)
OUTPUT: Peak Displacement [mm]
Peak Stress [MPa]
Note: Response at the middle node of the beam (node no 7) for the 1st mode frequency f=42.49[Hz]

Peak Displacement [mm]			Peak Stress [MPa]		
NAFEMS	Robot	**Difference**	NAFEMS	Robot	**Difference**
13.45	13.43	0.15%	241.9	242.4	0.21%

Results of Periodic Forced Vibration (5P)

Forcing function: F=Fo[sin(2πft)-sin(3(2πft))]
OUTPUT: Peak Displacement [mm]
Peak Stress [MPa]
Note: Response at the middle node of the beam (node no 7) for the frequency f=20[Hz]

Peak Displacement [mm]			Peak Stress [MPa]		
NAFEMS	Robot	**Difference**	NAFEMS	Robot	**Difference**
0.951	0.954	0.32%	17.10	17.36	1.52%

Results of Impulse Forced Vibration (5T)

Forcing function: F=Fo
OUTPUT: Peak Displacement [mm] and the corresponding Time [s]
Peak Stress [MPa]
Static Displacement [mm]
Note: Response at the middle node of the beam (node no 7)

Peak Displacement [mm]			at the Time [s]		
NAFEMS	Robot	**Difference**	NAFEMS	Robot	**Difference**
1.043	1.047	0.38%	0.0117	0.0117	0.0%

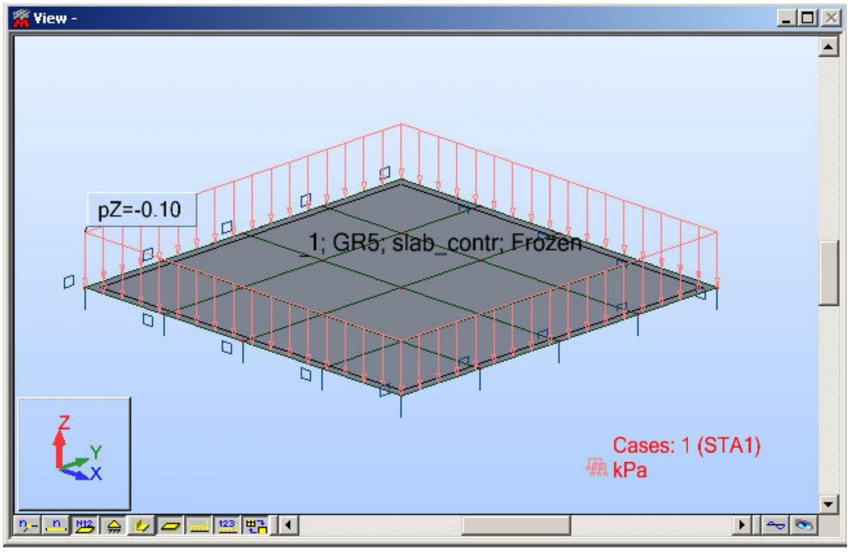
Peak Stress [MPa]			Static Displacement [mm]		
NAFEMS	Robot	**Difference**	NAFEMS	Robot	**Difference**
18.76	18.72	0.21%	0.538	0.537	0.19%

VERIFICATION EXAMPLE

TESTS No 13 - Vibrations of Simply Supported Thin Plate

Name of the test:	13, 13H, 13P, 13T
Reference:	NAFEMS R0016
Specification:	Dynamic analysis of an elastic plate

GEOMETRY: Length: A = B = 10 m
Thickness: t = 0,05 m



DATA DEFINITION:

Loading:	Uniform planar load Fo=100 [N/m^2] (Fo=0,1 [kN/m^2])	
Boundary condition:	X=Y=RZ=0	(at all nodes - Plate)
	Z=0	(at all edges)
	RX=0	(along edges X=0 & X=10 m)
	RY=0	(along edges Y=0 & Y=10 m)
Material properties:	E=200x10^9 N/m^2 ; ν=0,3 ; ρ=8000 kg/m^3	
Element type:	4-node quadrilateral shell elements (three models of mesh considered: 4x4, 8x8, and 16x16 elements).	

RESULTS COMPARISON:

Results of Modal Analysis (13)

OUTPUT: Frequencies [Hz]

Modes	NAFEMS	Robot (meshing 4x4)	Difference
1	2.377	2.512	5.68%
2&3	5.942	7.071	19.00%
4	9.507	11.738	23.47%
5&6	11.884	16.559	39.34%
7&8	15.449	21.311	37.94%

Modes	NAFEMS	Robot (meshing 8x8)	Difference
1	2.377	2.410	1.39%
2&3	5.942	6.216	4.61%
4	9.507	10.047	5.68%
5&6	11.884	13.201	11.08%
7&8	15.449	17.073	10.51%

Modes	NAFEMS	Robot (meshing 16x16)	Difference
1	2.377	2.385	0.34%
2&3	5.942	6.009	1.13%
4	9.507	9.638	1.38%
5&6	11.884	12.201	2.67%
7&8	15.449	15.840	2.53%

Results of Harmonic Forced Vibration (13H)

Forcing function: F=Fosin(2πft)

OUTPUT: Peak Displacement [mm]
Peak Stress [MPa]

Note: Response at the centre of the plate (node no 1) for the 1st mode frequency f=2.512[Hz] (4x4); f=2.410[Hz] (8x8); f=2.385[Hz] (16x16)

Meshing	Peak Displacement [mm]			Peak Stress [MPa]		
	NAFEMS	Robot	Difference	NAFEMS	Robot	Difference
4x4	45.42	44.66	1.67%	30.03	33.20	10.56%
8x8	45.42	45.11	0.68%	30.03	32.23	7.33%
16x16	45.42	45.11	0.68%	30.03	31.90	6.23%

Results of Periodic Forced Vibration (13P)

Forcing function: F=Fo[sin(2πft)-sin(3(2πft))]

OUTPUT: Peak Displacement [mm]
Peak Stress [MPa]

Note: Response at the centre of the plate (node no 1) for frequency f=1.2[Hz]

Meshing	Peak Displacement [mm]			Peak Stress [MPa]		
	NAFEMS	Robot	Difference	NAFEMS	Robot	Difference
4x4	2.863	3.070	7.23%	2.018	2.322	15.06%
8x8	2.863	2.915	1.82%	2.018	2.118	4.96%
16x16	2.863	2.884	0.73%	2.018	2.076	2.87%

Results of Impulse Forced Vibration (13T)

Forcing function: F=Fo

OUTPUT: Peak Displacement [mm] and the corresponding Time [s]
Peak Stress [MPa]
Static Displacement [mm]

Note: Response at the centre of the plate (node no 1)

Meshing	Peak Displacement [mm]			at the Time [s]		
	NAFEMS	Robot	Difference	NAFEMS	Robot	Difference
4x4	3.523	3.447	2.16%	0.210	0.200	4.76%
8x8	3.523	3.479	1.25%	0.210	0.210	0.0%
16x16	3.523	3.448	2.13%	0.210	0.210	0.0%

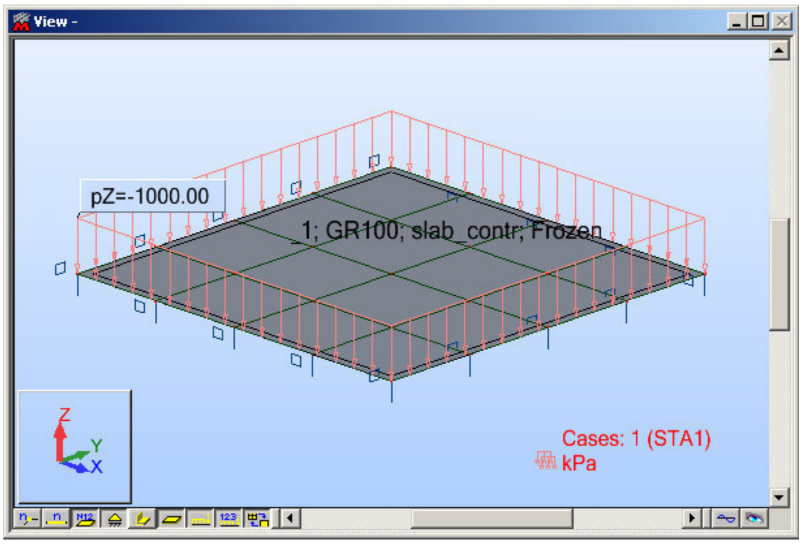
Meshing	Peak Stress [MPa]			Static Displacement [mm]		
	NAFEMS	Robot	Difference	NAFEMS	Robot	Difference
4x4	2.484	2.322	6.52%	1.817	1.767	2.75%
8x8	2.484	2.462	0.86%	1.817	1.774	2.37%
16x16	2.484	2.361	4.95%	1.817	1.775	2.31%

VERIFICATION EXAMPLE

TESTS No 21 - Vibrations of Simply Supported Thick Plate

Name of the test: 21, 21H, 21P, 21T
Reference: NAFEMS R0016
Specification: Dynamic analysis of an elastic plate

GEOMETRY: Length: A = B = 10 m
Thickness: t = 1,0 m



DATA DEFINITION:

Loading:	Uniform planar load Fo=10^6 [N/m^2] (Fo=1000 [kN/m^2])	
Boundary condition:	X=Y=RZ=0	(at all nodes - Plate)
	Z=0	(at all edges)
	RX=0	(along edges X=0 & X=10 m)
	RY=0	(along edges Y=0 & Y=10 m)
Material properties:	E=200x10^9 N/m^2 ; ν=0,3 ; ρ=8000 kg/m^3	
Element type:	4-node quadrilateral shell elements (three models of mesh considered: 4x4, 8x8, and 16x16 elements).	

RESULTS COMPARISON:

Results of Modal Analysis (21)

OUTPUT: Frequencies [Hz]

Modes	NAFEMS	Robot (meshing 4x4)	Difference
1	45.897	48.589	5.87%
2&3	109.44	129.99	18.78%
4	167.89	207.34	23.50%
5&6	204.51	280.29	37.05%
7&8	256.50	350.94	36.82%

Modes	NAFEMS	Robot (meshing 8x8)	Difference
1	45.897	46.542	1.41%
2&3	109.44	114.67	4.78%
4	167.89	178.12	6.09%
5&6	204.51	226.35	10.68%
7&8	256.50	283.72	10.61%

Modes	NAFEMS	Robot (meshing 16x16)	Difference
1	45.897	46.065	0.37%
2&3	109.44	110.94	1.37%
4	167.89	170.89	1.79%
5&6	204.51	210.56	2.96%
7&8	256.50	264.31	3.04%

Results of Harmonic Forced Vibration (21H)

Forcing function: F=Fosin(2πft)

OUTPUT: Peak Displacement [mm]
Peak Stress [MPa]

Note: Response at the centre of the plate (node no 1) for the 1st mode frequency f=48.589[Hz] (4x4); f=46.542[Hz] (8x8); f=46.065[Hz] (16x16)

Meshing	Peak Displacement [mm]			Peak Stress [MPa]		
	NAFEMS	Robot	Difference	NAFEMS	Robot	Difference
4x4	58.33	58.93	1.03%	800.8	866.9	8.25%
8x8	58.33	59.64	2.25%	800.8	825.6	3.10%
16x16	58.33	59.71	2.37%	800.8	807.6	0.85%

Results of Periodic Forced Vibration (21P)

Forcing function: F=Fo[sin(2πft)-sin(3(2πft))]

OUTPUT: Peak Displacement [mm]
Peak Stress [MPa]

Note: Response at the centre of the plate (node no 1) for frequency f=20[Hz]

Meshing	Peak Displacement [mm]			Peak Stress [MPa]		
	NAFEMS	Robot	Difference	NAFEMS	Robot	Difference
4x4	4.929	6.127	24.31%	67.67	91.19	34.76%
8x8	4.929	5.328	8.09%	67.67	75.23	11.17%
16x16	4.929	5.175	4.99%	67.67	71.40	5.51%

Results of Impulse Forced Vibration (21T)

Forcing function: F=Fo

OUTPUT: Peak Displacement [mm] and the corresponding Time [s]
Peak Stress [MPa]
Static Displacement [mm]

Note: Response at the centre of the plate (node no 1)

Meshing	Peak Displacement [mm]			at the Time [s]		
	NAFEMS	Robot	Difference	NAFEMS	Robot	Difference
4x4	4.524	4.563	0.86%	0.0108	0.0104	3.7%
8x8	4.524	4.528	0.09%	0.0108	0.0105	2.78%
16x16	4.524	4.578	1.19%	0.0108	0.0105	2.78%

Meshing	Peak Stress [MPa]			Static Displacement [mm]		
	NAFEMS	Robot	Difference	NAFEMS	Robot	Difference
4x4	62.11	66.32	6.78%	2.333	2.317	0.69%
8x8	62.11	59.86	3.62%	2.333	2.323	0.43%
16x16	62.11	60.25	2.99%	2.333	2.322	0.47%

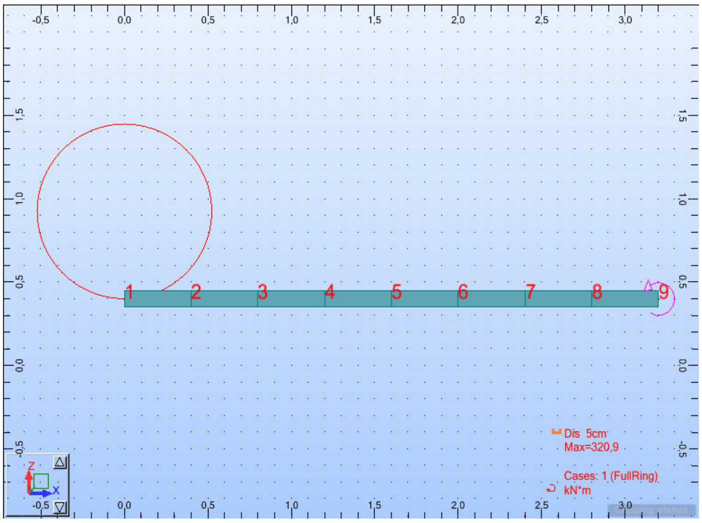
LARGE ROTATIONS
AND
DISPLACEMENTS

VERIFICATION EXAMPLE

TEST GNL-5 – Large rotations and displacements of a straight cantilever

Name of the test:	GNL-5
Reference:	NAFEMS R0065
Specification:	Geometric nonlinearity

GEOMETRY: Length: 3,2 m
Cross section: rectangular 0,1x0,1 m

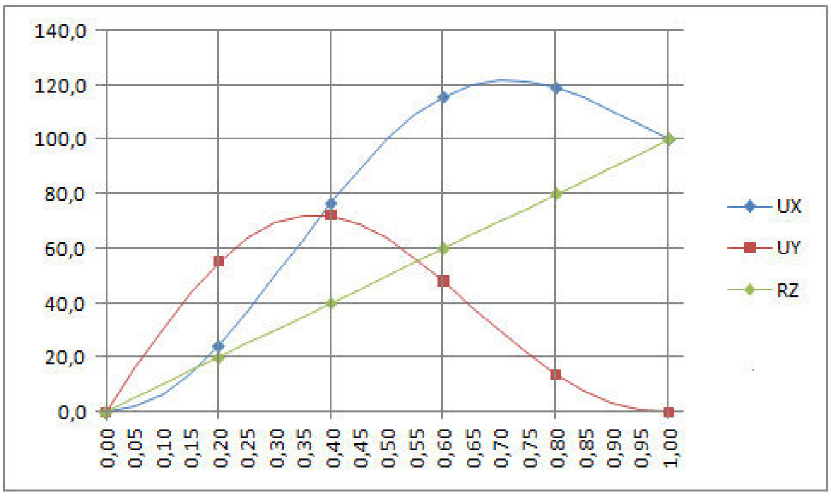


DATA DEFINITION:

Loading:	Concentrated moment at the end point M = 3 436,1 [kN*m] (applied in 36 equal increments)
Boundary condition:	Built-in at the begin
Material properties:	E=210x10^9 N/m^2 ; ν=0,0 (density not considered)
Discretization:	8 beam elements, 40 load increments, analysis P-delta, nonlinear.

RESULTS COMPARISON:

"Dimensionless" value of tip displacements and rotations (UX=100*ux/L, UY=100*uy/L, RZ=100*rz/2*π) are presented on the analytical plot, and summarized in the table, below.



Normalized horizontal displacement at tip versus normalized bending moment

M/Mmax	***analyt.***	**Robot**	***diff***
0,2	24,3	24,3	-0,3%
0,4	76,6	76,5	-0,1%
0,6	115,6	115,8	0,1%
0,8	118,9	119,3	0,3%
1,0	100,0	100,0	0,0%

Normalized vertical displacement at tip versus normalized bending moment

M/Mmax	***analyt.***	**Robot**	***diff***
55,0	55,0	55,0	0,1%
72,0	72,0	72,3	0,4%
48,0	48,0	48,4	0,9%
13,7	13,7	14,0	1,6%
0,0	0,0	0,0	0,0%

Normalized rotations at tip versus normalized bending moment

M/Mmax	***analyt.***	**Robot**	***diff***
0,0	20,0	20,0	0,0%
0,0	40,0	40,0	0,0%
0,0	60,0	60,0	0,0%
0,0	80,0	80,0	0,0%
0,0	100,0	100,0	0,0%

CONCLUSIONS

The results and accuracy achieved in verification examples confirm the quality and reliability of Robot. This state-of-the-art structural analysis and design software gives sufficient accuracy limited only by the precision of modeling.