10.1515/tvsb-2015-0015

Transactions of the VŠB – Technical University of Ostrava

Civil Engineering Series, No. 2, Vol. 15, 2015

paper #15

Maciej MAJOR[1], Krzysztof KULIŃSKI[2]

COMPARATIVE NUMERICAL ANALYSIS OF ADVERTISING BOARD TOWER USING ADINA AND AUTODESK ROBOT STRUCTURAL ANALYSIS

Abstract

In this paper the subject of comparative stress and displacement analysis for different computer aided design program environments is discussed. For the purposes of analysis ADINA and Autodesk ROBOT program were used. In both programs an advertising board tower was modeled. Static analysis was adopted for the computations. Obtained results allowed showing small differences and limitations between those two environments.

Keywords

Finite Element Method, FEM, Comparative analysis, ADINA, Autodesk ROBOT Structural Analysis.

1 INTRODUCTION

Stresses and displacements are fundamental values which have to be checked at the early stage of designing almost every structure. According to guidelines provided with European Eurocodes or national standards all limit states have to be qualified for all elements of structure. When those two limit states are qualified it is assumed that construction is well-defined. There are two main types of mentioned limit states in accordance with [1]:

- ULS (Ultimate Limit State) – comparison between the real stress values of analyzed structure to the permissible stress,
- SLS (Serviceability Limit State) – comparison between the real displacement value of element to the permissible displacement.

It can be stated that the Ultimate Limit State is the most important during design of any elements because there is a necessity to check out all the requirements showed in standards. Same conditions are applied to the check-out of Serviceability Limit State. Normally computations are carried out in the following order – check out ULS than SLS. In most cases this order is sufficient but in some, this order has to be inverted. That can occur mainly when an element meets all the requirements of ULS, but displacements or deflections have higher value than the acceptable range provided by standards.

[1] Ing. Maciej Major, Ph.D., Department of Technical Mechanics, Faculty of Civil Engineering, Częstochowa University of Technology, Akademicka 3, 42-200 Częstochowa, Poland, phone: (+48) 34 3250965, e-mail: mmajor@bud.pcz.czest.pl.

[2] Ing. Krzysztof Kuliński, MSc.., Department of Mechanics and Machine Design Foundation, Faculty of Mechanical Engineering and Computer Science, Częstochowa University of Technology, Armii Krajowej 21, 42-201 Częstochowa, Poland, phone: (+48) 34 3250654, e-mail: krzysztku@gmail.com.

Technological progress both in civil engineering and computer science led into some kind of revolution in solving structure problems. There is no longer need of significant simplifies in an adopted model for decreasing the amount of time necessary to make computations by one or group of engineers. Nowadays computers and wide range of available computer-aided design software allow users to make complex numerical analysis even in household conditions or small design office. In order to make correct computations of an element or structure (even in three-dimensional space) providing almost the same behavior as in reality, detailed knowledge of solved problem and software environment is required. In that case there is no need to simplify construction to basic schemes where one scheme depends on the results from the other one. If construction is divided to basic schemes change of any element in any scheme may cause necessity of resolve the whole problem again. In many cases there is a situation, where the amount of used elements in construction, its static scheme or complexity makes the analytical solution of stated problem impossible to obtain by known methods. Moreover dividing construction into the basic parts, results in increased time necessary to obtain solution and increased risk of making mistake in computations which have the influence on final results.

Due to the fact that numerical methods have many advantages in computing various types of structures, these methods are willingly used. There is a vast number of publications affecting the topic of finite element method usage in civil engineering and mechanics. Čajka and Krejsa [2] validated the computational model with load test carried out on the real construction. The computational model was utilized to check if repaired rooflight steel structure was free of permanent deformations. Computations with finite element method of castellated beam with imperfections were shown in [3]. Authors in this paper attempted to determine the coefficients of lateral torsional buckling and describe the stress state of concerned beam profile. The comparative analysis of the real deformation values to the finite element model results for plate located in subsoil was shown in [4]. The experimental and numerical analysis of steel joints with physical and geometrical nonlinearities was presented in [5]. Lausová et al. [6] compared the results obtained from numerical modeling in ANSYS with experimental results in order to verify statically indeterminate behavior of steel frame construction exposed to high temperatures. Koktan and Brožovsky [7] proposed an implementation of creep analysis for reinforced concrete structures and the direct stiffness method for reinforced concrete frames. Authors presented their solution with comparison to the EN 1992-1-1 technical standard. Numerical modeling not only concerns the computations of structure strength and deformations but it can be also helpful in determine the structure reliability. Numerical probabilistic reliability assessment of truss construction using MATLAB and Monte Carlo simulation technique was presented in [8]. Direct optimized probabilistic calculation which is a pure numerical approach without any simulation technique required was discussed in [9].

In this paper the comparative analysis of obtained displacement and stress results between two different finite element method based programs – ADINA and Autodesk ROBOT Structural Analysis is discussed. The comparative analysis between ADINA and ROBOT software has also been discussed in [10] containing bending problem of two-dimensional beam. Comparative analysis in ADINA software was also made in [11, 12]. Model adopted for the comparative analysis in this paper represents a typical cantilever advertising board tower which can be spotted in many cities in Poland near the main roads. Numerical model includes main steel pipe column, where bottom surface has all displacements and rotations deactivated, several steel pipe beams which are connected to the main column supporting the aluminium board. It was also assumed that all dimensions were ideal – there are no geometry imperfections. All computations were limited only to the static analysis. Steel and aluminum are assumed as linear, elastic and isotropic material models. Detailed technical data concerning examined scheme are presented in section 3 in this paper.

2 COMPARED PROGRAM CHARACTERISTICS

ADINA – (Automatic Dynamic Incremental Nonlinear Analysis) – software based on finite element method. Numerical models can be designed in two-dimensional or in three-dimensional space. This software allows numerical computations in statics, dynamics, also crack propagation analysis, crack mechanics, flow of liquids and gases, acoustic wave propagation, thermal radiation and more. Moreover it is possible to connect two different analyses and compute them at the same time – for example computations including dynamic effects caused by liquid flow. Additionally there can also be done geotechnical, biomechanical, electromagnetic and collapse analyses. Almost all known material models can be implemented for analyzed structure if only proper physical properties are known.

Autodesk ROBOT Structural Analysis – software produced mainly for civil and mechanical engineering usage. The program is subdivided into sixteen basic modules which allow user to choose the type of element or construction and a coordinate system (two-dimensional or three-dimensional space). Some modules have predefined assumptions, for example planar trusses have predefined hinges in all nodes and user does not have to declare them on its own. Shell and volume elements are computed using Finite Element Method. In shell elements user can choose from 3-node triangular or 4-node quadrilateral finite elements and for volume analysis 4-node tetrahedral and 8-node hexahedral finite elements. There can be also adopted emitter elements which help increase mesh density in particular areas. Moreover software include base of predefined profiles used around the world, tools which help generate typical constructions and special modules designed for computing elements with requirements specified in current national standards and European Eurocodes.

3 NUMERICAL MODEL

For the numerical solution both in ADINA and ROBOT an advertising board tower in three-dimensional space was adopted. Profiles of column and supporting beams are made of steel pipes in accordance with Polish national standard (see [13, 14]). The complex structure of an advertising board was simplified to the volume with constant thickness of 5.00 cm. It was assumed that the advertising board is made of aluminum material. For the bottom surface of pipe column all displacements and all rotations were disabled (clamped support). There are two loads acting in the model – the wind pressure load on the board and dead load covering the whole construction. Wind pressure of 1630 Pa was calculated in accordance with [15], dead load with [16]. Following model assumptions were adopted (Tab.1):

Tab.1: Material properties

	Material		Unit
	Steel	**Aluminium**	
Type:	S235JR	3150-H14	-
Young modulus	210	70	GPa
Poisson ratio	0.30	0.33	-
Density	7860	2700	kg/m^3
Material model	linear elastic isotropic	linear elastic isotropic	-

Dimensions of the analyzed structure are as follows (Fig.1-3):

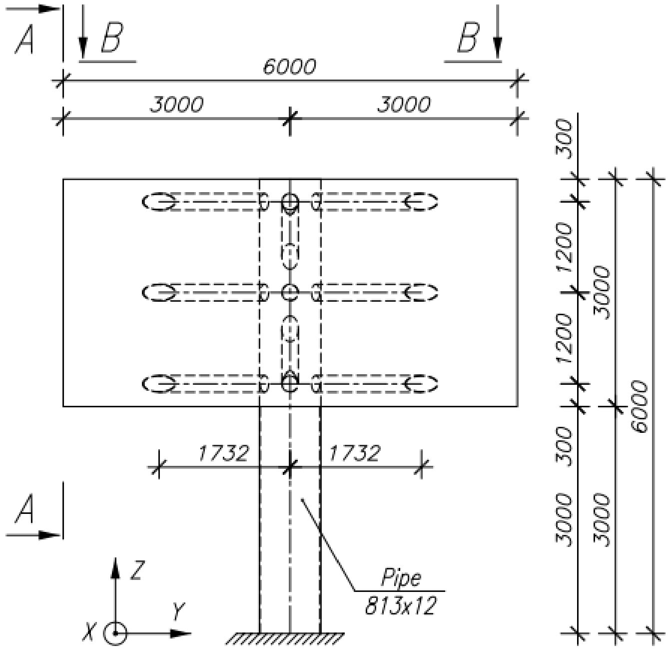


Fig. 1: Front view on the advertising board tower

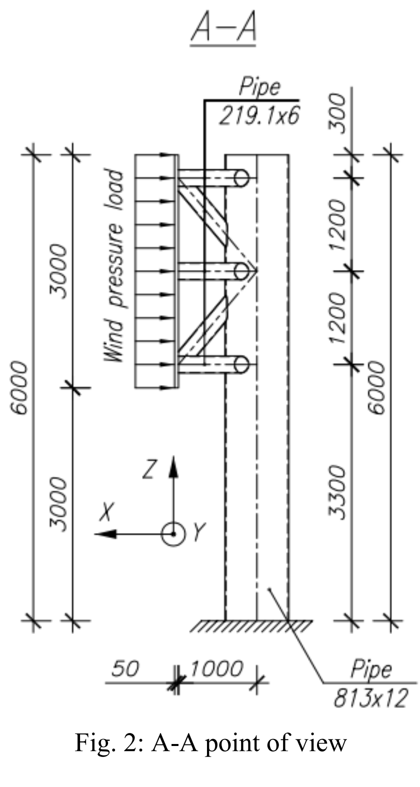


Fig. 2: A-A point of view

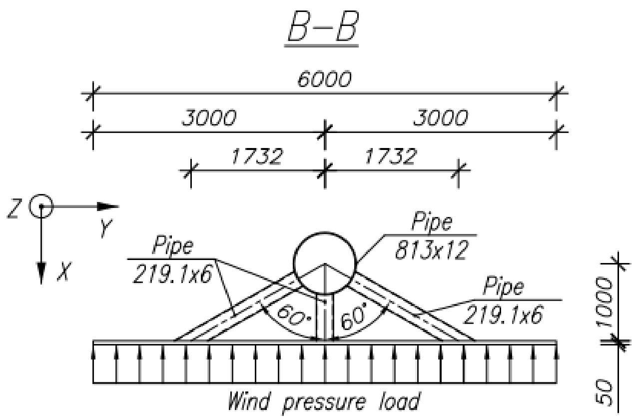
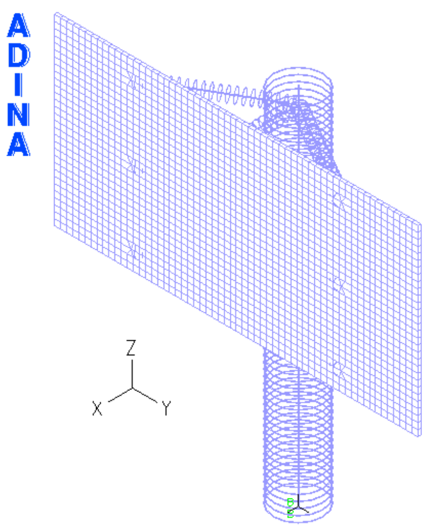


Fig. 3: B-B point of view

In the ADINA program the whole model was prepared in three-dimensional space (Fig. 4a). The advertising board was meshed with 4-node quadrilateral "shell" finite elements [17, 18]. The column and supporting beams were meshed with 2-node "beam" finite elements. Connection between supporting beams and the main column and also between mentioned beams and the advertising board tower was assumed as rigid. Bottom surface of the column pipe has all degrees of freedom fixed. The whole model contains 2088 nodes and 2010 finite elements, where the board consists of 1800 shell elements, supporting beams contain 150 beam elements and the main column consists of 60 elements.

In the ROBOT also the advertising board was meshed with 4-node quadrilateral shell finite elements, while board support pipes were described as beam elements and main pipe as a column element. The board table contains 2061 nodes and 1800 finite elements (Fig. 4b).

a)



b)

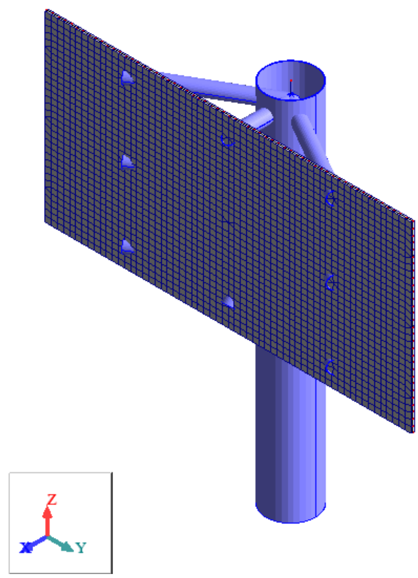


Fig. 4: Numerical model of the advertising board tower – a) ADINA, b) ROBOT

4 RESULTS

In this chapter there are presented obtained results from the numerical analysis. The comparison of displacements and stresses is subdivided into two groups. The first group contains displacements and stresses comparison for the advertising board, whereas the second one contains comparison of displacements and effective stress (Huber and Mises) for the steel main pipe column. Example of effective stress band plot in ADINA was shown in Fig. 5, while displacements in X direction for the advertising board in ROBOT software was shown in Fig. 6. The points where stresses and displacements were measured both in ADINA and ROBOT were shown in Fig. 7. Stresses on the board were measured in the shell midsurface in both programs.

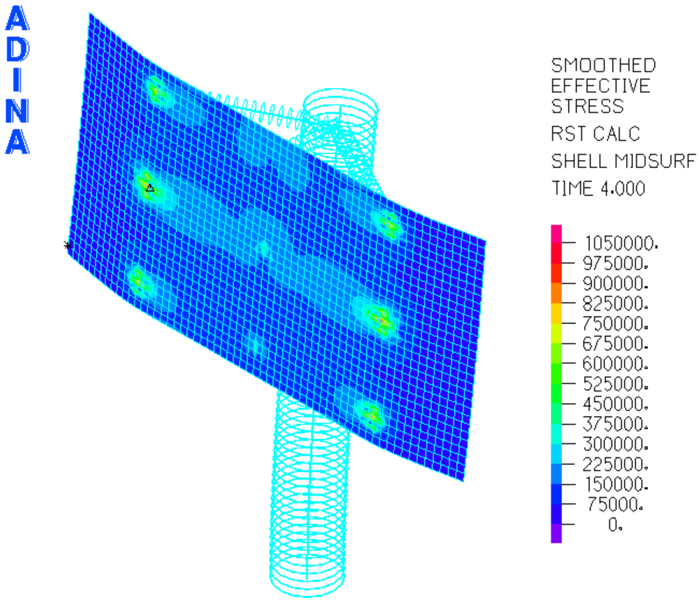


Fig. 5: Effective stress (Huber and Mises) band plot with deformation for the advertising board in ADINA. Isometric point of view

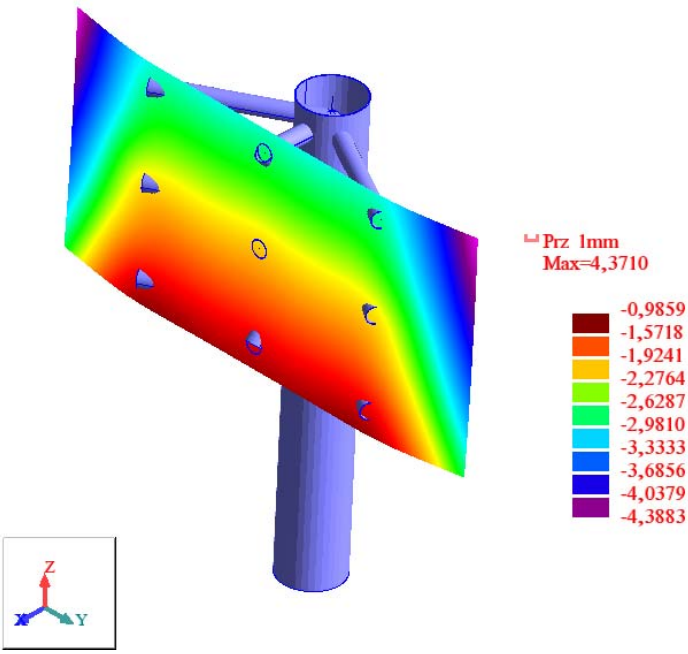


Fig. 6: X-axis displacements band plot with deformation for the advertising board in ROBOT. Isometric point of view

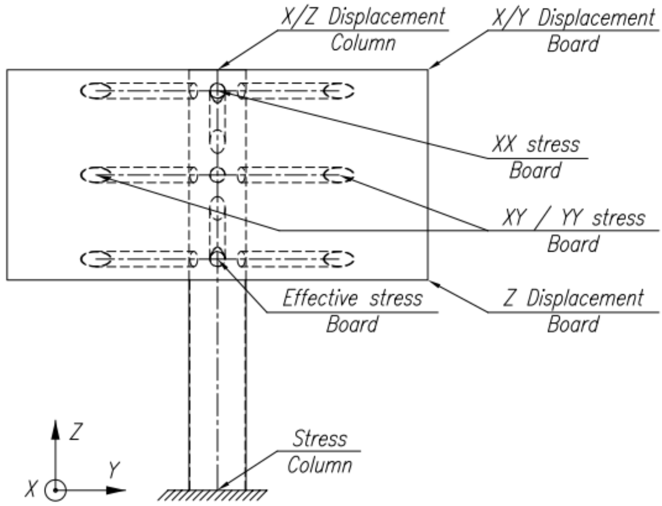


Fig. 7: View of compared points in Ultimate and Serviceability Limit State
both in ADINA and ROBOT software

Tab. 2: Stress and displacement results for the advertising board

Board					
		Unit	**ADINA**	**ROBOT**	**Percentage difference**
Displacement	**X**	**mm**	-2.8260	-2.8523	0.93%
	Y	**mm**	±0.00669	±0.00702	4.93%
	Z	**mm**	-0.3083	-0.3036	1.55%
Stress	**Effective**	**MPa**	1.1482	1.1953	4.10%
	XX	**MPa**	1.2965	1.2302	5.39%
	XY	**MPa**	0.4709	0.4769	1.11%
	YY	**MPa**	1.0201	1.0314	1.27%

Tab. 3: Stress and displacement results for the main pipe column

Column					
		Unit	**ADINA**	**ROBOT**	**Percentage difference**
Displacement	**X**	**mm**	-1.7730	-1.7908	0.99%
	Z	**mm**	-0.0280	-0.0274	2.14%
Stress	**Effective**	**MPa**	29.1400	29.4950	1.22%

"-" signs in the displacements (see Tab. 2 and Tab. 3) means that the values are in opposite direction than adopted coordinate system. "-" signs in the stresses means compressing stress, the "+" sign means tensile stress.

5 CONCLUSION

Preparation of identical models and performing numerical analysis both in ADINA and ROBOT allowed comparing the obtained results. Results for the analyzed advertising board tower in ADINA and ROBOT are given in Tab. 2 and Tab. 3. The percentage differences were calculated as an absolute difference between programs. Comparing computed displacements for the board (see Tab. 2), it can be seen that maximum difference is in Y-axis equal 4.93%. The displacements in X-axis, which are the most important from the engineering point of view for this problem are almost identical showing good agreement between programs. Stresses in the board (see Tab. 2) which were measured in shell midsurfaces have slightly higher differences. The difference between effective stress in ADINA and ROBOT is 4.10% and 5.39% in XX stress, respectively. The XY and YY stresses show good agreement. Comparing the results of displacements and stresses for the main steel column absolute difference between programs varies from 0.99% to 2.14%. Moreover to validate the numerical results stress for the clamped end in steel column had been determined analytically. The analytical value of stress in clamped end was equal 29.0272 MPa. Comparing analytical stress value to the result obtained in ROBOT, the absolute error stands at 0.39%, whereas the stress result obtained from ADINA to the analytical result stands at 1.61%. Due to the fact that differences between programs were not greater than 5.39% and the difference between analytical stress in clamped end to the numerical results were around 1% numerical models can be treated as well reflecting the real behaviour. The differences between programs are mainly connected with different solvers which were originally implemented, cause utilized mesh size and type of elements were identical. It should be noted that there is no proper method excluding FEM or laboratory research on scaled model to obtain result close to the reality.

The significant influence on obtained results have not only the mesh size but also the type of adopted finite elements in ADINA. There are three groups of elements in ADINA depended on performed analysis. For one-dimensional problems there are finite elements with a shape of line, in two-dimensional problems there are triangular and quadrilateral finite elements and in three-dimensional tetrahedral and hexahedral elements. Those two-dimensional and three-dimensional finite elements are also subdivided into subgroups where they have different number of nodes greater than minimal necessary to create a shape of chosen element. In some cases in two or three-dimensional analysis a problem can occur with triangle or tetrahedral elements with minimal necessary number of nodes (3 or 4-node elements) causing displacement lock [17]. Due to this fact 4-node or more, quadrilateral or 8-node or more hexahedral elements should be used. Using higher order elements with more nodes prevent from displacement locking, results have better quality but analysis became very costly - time and required computer virtual memory necessary for performing analysis increases drastically [19, 20].

Both programs are good assistance for engineers due to its calculation speed for especially complex structures. The ROBOT program due to its intuitive environment and association with national construction standards is commonly used in design offices around the world. It allows user not only to calculate the forces in structure but also calculate required reinforcement of beams, slabs, wall etc. There can also be performed modal, seismic, spectral analysis etc. Moreover in the ROBOT can be designed and calculated connections between steel, timber and concrete elements in accordance with national standards. In the ADINA there are no such modules where element is calculated in accordance with national standard, but program allow users to perform complex and detailed analysis of any element or construction with finite element method. For that instance engineer must know if displacements, stresses etc. are not exceeding the acceptable value. The ADINA program allow solving problems not only concerning civil engineering but also problems from other research areas. Almost every mechanical problem can be solved with finite element method. Both programs in hands of experienced engineer gives measurable benefits in saved time required to perform analysis of complex structures and also allow user to have a wider point of view on element or construction behaviour.

ACKNOWLEDGMENT

The contribution has been created with the financial support from the budget of the MoravianSilesian Region within the frame of program Backing of the science and research in the Moravian-Silesian Region, number of the program: RRC/07/2014.

LITERATURE

[1] PN-EN 1990-1-1:2004. *Eurokod - Podstawy projektowania konstrukcji*. Warszawa: PKN, 2004.

[2] ČAJKA R. & KREJSA M. Validating a computational model of a rooflight steel structure by means of a load test, *Applied Mechanics and Materials*, 2014, DI - DIV, pp. 592-598, ISBN: 978-3-03835-005-7.

[3] MIKOLÁŠEK D., SUCHARDA O. & BROŽOVSKY J. Numerical analysis of castellated beam, *Transactions of the VŠB - Technical University of Ostrava, Civil Engineering Series*. Volume 13, Issue 2, 2013, Pages 98–104, ISSN (Online) 1804-4824, ISSN (Print) 1213-1962, DOI: 10.2478/tvsb-2013-0015, 2013.

[4] LABUDKOVÁ J. & ČAJKA R. Comparison of experimentally measured deformation of the plate on subsoil and the results of 3D numerical model, *Transactions of the VŠB - Technical University of Ostrava, Civil Engineering Series*. Volume 14, Issue 1, 2014, Pages 57–66, ISSN (Online) 1804-4824, DOI: 10.2478/tvsb-2014-0008, 2014.

[5] MIKOLÁŠEK D., LOKAJ A., BROŽOVSKY J. & SUCHARDA O., Experimental and numerical analysis of steel joints in round wood, *Transactions of the VŠB - Technical University of Ostrava, Civil Engineering Series*. Volume 14, Issue 2, 2014, Pages 1–10, ISSN (Online) 1804-4824, DOI: 10.2478/tvsb-2014-0027, 2014.

[6] LAUSOVÁ L., MATEČKOVÁ P. & SKOTNICOVÁ I. Experimental and numerical analysis of steel frame structure exposed to high temperature, *Transactions of the VŠB - Technical University of Ostrava, Civil Engineering Series.* Volume 14, Issue 2, 2014, Pages 64–68, ISSN (Online) 1804-4824, DOI: 10.2478/tvsb-2014-0022, 2014.

[7] KOKTAN J. & BROŽOVSKY J. Numerical modelling of time-dependent behaviour of reinforced concrete structure with use of B3 model, *Transactions of the VŠB - Technical University of Ostrava, Civil Engineering Series*. Volume 14, Issue 2, 2014, Pages 38–45, ISSN (Online) 1804-4824, DOI: 10.2478/tvsb-2014-0019, 2014.

[8] VAŠEK J. & KREJSA M. Probabilistic assessment of truss construction in Matlab software platform, *Transactions of the VŠB - Technical University of Ostrava, Civil Engineering Series*. Volume 14, Issue 1, 2014, Pages 100–109, ISSN (Online) 1804-4824, DOI: 10.2478/tvsb-2014-0013, 2014.

[9] KREJSA M., JANAS P. & ČAJKA R. Using DOProC method in structural reliability assessment, *Applied Mechanics and Materials*. Vols. 300 - 301, 2013, p. 860-869 (10 p). ISSN: 1662-7482. DOI: 10.4028/www.scientific.net/AMM.300-301.860.

[10] MAJOR, M. & MAJOR, I. Computer aided design – comparative analysis of widely available software with analytical method. *Transactions of the VŠB - Technical University of Ostrava, Civil Engineering Series*. Volume 14, Issue 2, 2014, Pages 79–84, ISSN (Online) 1804-4824, DOI: 10.2478/tvsb-2014-0023, 2014.

[11] MAJOR, M. & MAJOR, I. Comparative analysis of the distribution of effective stress in Mooney and Zahorski materials using ADINA Software. *Advanced Materials Research.* 2014, MXX, pp. 165-170. ISSN: 1662-8985.

[12] MAJOR, I. Numerical analysis of wave phenomena in hyperelastic Mooney – Rivlin and Zahorski materials. *Civil and environmental engineering, Scientific - Technical Journal*. 2014, X, Nr. 1, pp. 42-48. ISSN: 1336-5835.

[13] PN-EN 10210-2: 2000. *Kształtowniki zamknięte wykonane na gorąco ze stali konstrukcyjnych niestopowych i drobnoziarnistych -- Tolerancje, wymiary i wielkości statyczne*. Warszawa: PKN, 2000.

[14] PN-EN 10219-2:2000. *Kształtowniki zamknięte ze szwem wykonane na zimno ze stali konstrukcyjnych niestopowych i drobnoziarnistych -- Tolerancje, wymiary i wielkości statyczne*. Warszawa: PKN, 2000.

[15] PN-EN 1991-1-4:2008. *Eurokod 1: Oddziaływania na konstrukcje. Część 1-4: Oddziaływania ogólne - Oddziaływania wiatru*. Warszawa: PKN, 2008.

[16] PN-EN 1991-1-1:2004. *Eurokod 1: Oddziaływania na konstrukcje. Część 1-1: Oddziaływania ogólne. Ciężar objętościowy, ciężar własny, obciążenia użytkowe w budynkach*. Warszawa: PKN, 2004.

[17] BATHE, K. J. *Finite Element Procedures*. 1st ed. Cambridge MA : Klaus Jürgen Bathe, 2006. 1037 pp. ISBN 9780979004902.

[18] BATHE, K. J. Nonlinear Finite Element Analysis and ADINA. In *Proceedings of the 9th ADINA Conference*. Cambridge, MA, 1993, pp. 511-891. ISSN: 0045-7949.

[19] ZIENKIEWICZ, O. C. *Metoda elementów skończonych.* 1st ed. Warszawa: Arkady, 1972. 455 p.

[20] ZIENKIEWICZ, O. C. & TAYLOR, R. L. *The Finite Element Method. Volume 2: Solid Mechanics.* 1st ed. Oxford : Butterworth-Heinemann, 2000. 438 pp. ISBN 0-7506-5055-9.

Reviewers:

Doc. Ing. Ivan Němec, CSc., Institute of Structural Mechanics, Faculty of Civil Engineering, Brno University of Technology, Czech Republic.

Ing. Mikolášek David, Ph.D., Department of Structural Mechanics, Faculty of Civil Engineering, VŠB – Technical University of Ostrava, Czech Republic.