10.1515/tvsb-2017-0027

Transactions of the VŠB – Technical University of Ostrava

Civil Engineering Series, Vol. 17, No. 2, 2017

paper #27

Maciej MAJOR[1], Izabela MINDA[2], Krzysztof KULIŃSKI[3], Izabela MAJOR[4]

COMPARATIVE NUMERICAL ANALYSIS OF OBTAINED STRESS AND DISPLACEMENT RESULTS FROM FEM PROGRAMS - ROBOT STRUCTURAL ANALYSIS AND ADINA IN EXAMPLE OF A STEEL FOOTBRIDGE SUBJECTED TO THE DEAD LOAD

Abstract

In this paper the comparative analysis of obtained stress and displacements results from two different FEM programs – Robot Structural Analysis and ADINA was performed. As a subject of the analysis, the numerical model of the arch footbridge in steel construction was adopted.

Keywords

Steel footbridge, arch footbridge, statics analysis, ADINA, Autodesk ROBOT.

1 INTRODUCTION

Over the years, the necessity of crossing obstacles such as watercourses precluding or significantly impeding the possibility of getting to the specific location led engineers to create initially simple structures providing safe passage for people. Nowadays mentioned necessity was further compounded via intensive road network development, thus the realization of increasing number of footbridges significantly eased communication in large cities, providing comfort and safety for both pedestrians and vehicles.

Basic footbridges form of design includes: single span or multi-span beam bridge, truss bridge, suspended bridge and arch bridge [1]. Selection of appropriate type of construction depends on planned span dimension, its function (type of obstacles, the form of utilization, location etc.), time of realization, project economics, etc. There are many researches of bridge structures (also existing) which allows to designate competent solutions [2,3].

Through arch construction utilization in steel footbridges, which was the subject of this paper, it is possible to construct span reaching up to 80 meters [4]. Mentioned construction also allows the effective geometry utilization underlining the aesthetic appeal in the surrounding area.

The main purpose of performed comparative analysis was to collate stress and displacements values of the footbridge model in two different FEM programs. A similar comparison between those two softwares but for the advertising board tower was conducted in [5], whereas the comparison between widely available software and analytical methods the authors presented in article [6].

[1] Assoc. Prof. Eng. Maciej Major, Ph.D., Częstochowa University of Technology, Faculty of Civil Engineering, Department of Steel Constructions and Building Materials, Akademicka 3 street, 42-200 Częstochowa, Poland, phone: (+48) 34 3250 965, e-mail: mmajor@bud.pcz.czest.pl.

[2] Eng. Izabela Minda, Częstochowa University of Technology, Faculty of Civil Engineering (student), Akademicka 3 street, 42-200 Częstochowa, Poland, izabela.minda@wp.pl.

[3] MSc. Eng. Krzysztof Kuliński, Częstochowa University of Technology, Faculty of Civil Engineering, Department of Concrete Constructions and Geotechnics, Akademicka 3 street, 42-200 Częstochowa, Poland, kkulinski@bud.pcz.czest.pl.

[4] Assoc. Prof. Eng. Izabela Major, Ph.D., Częstochowa University of Technology, Faculty of Civil Engineering, Department of Technical Mechanics and Engineering Graphics, Akademicka 3 street, 42-200 Częstochowa, Poland, phone: (+48) 34 3250 901, e-mail: imajor@bud.pcz.czest.pl.

2 NUMERICAL MODELS ASSUMPTIONS

2.1 Computational model in Robot Structural Analysis software

The subject of the analysis - freely supported footbridge in steel construction was presented in Fig. 1. As a material, S355 steel was assumed. Functional width dimension was planned to give opportunity in free two directions movement. Moreover, it was assumed that the emergency vehicles are not allowed to pass through the passage. Numerical model prepared in the Autodesk Robot was presented in Fig. 1.

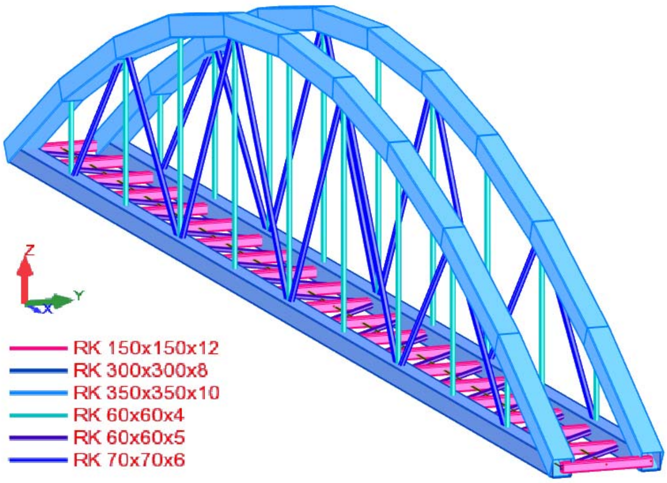


Fig.1: Perspective view of steel footbridge numerical model in Robot Structural Analysis software

2.2 Boundary conditions

Presented in Fig. 1. model of footbridge was assumed as simply supported. In point 1 and 2 (see Fig. 2) displacement along X-axis and rotation with respect to the Y-axis were allowed in relation to the presented coordinate system, whereas at points 3 and 4 only rotation with respect to the Y-axis was allowed, respectively.

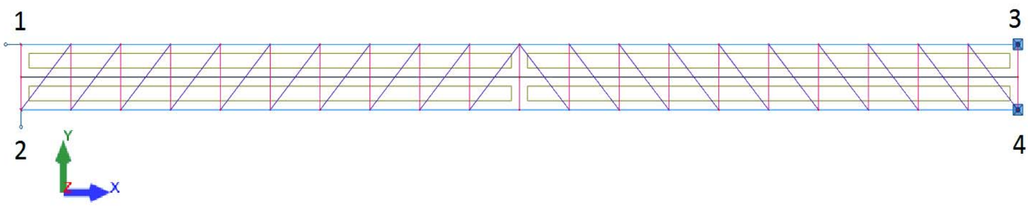


Fig.2: Specified points with adopted boundary conditions in Autodesk Robot

2.3 Model geometry

Analyzed construction comprise steel arch footbridge with span length equal 24 m and width between longitudinal external axes equal 1.4 m. The geometry of the arc was created as a segment of circle with radius R = 20 m cut off via chord equal to unsupported length of the girder beams. In order to provide additional stiffness of presented structure, bracings in YX and ZX plane were added. The whole model was made of square tubes, where cross-sections dimensions met the conditions of Ultimate Limit State. Beam elements type was used to model discussed footbridge in the ROBOT program. Performed calculations referencing to the Ultimate Limit State were presented in further section of this paper. The dimensions of the footbridge were presented in Figs. 3 and 4. In Tab. 1 types of the cross-section applied for each individual footbridge element were presented.

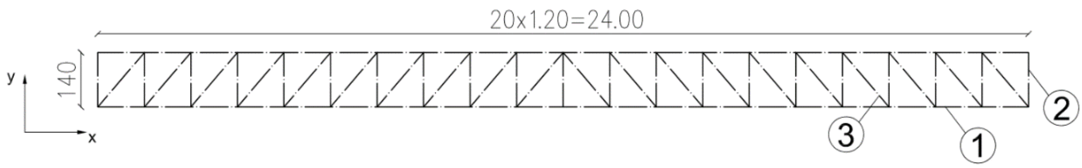


Fig.3: Design dimensions [m] of the footbridge. XY plane view

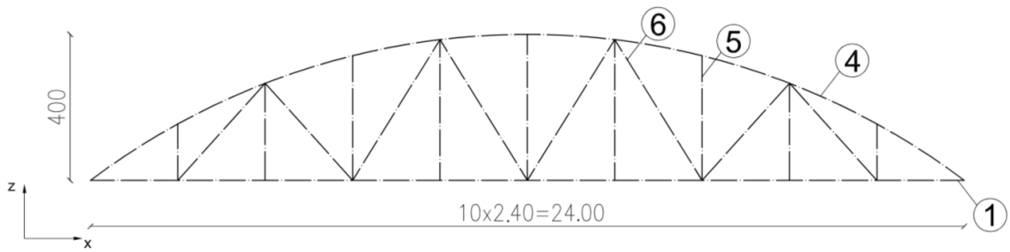


Fig.4: Design dimensions [m] of the footbridge. XZ plane view.

Tab.1: Cross-sections utilized in the construction. RK appendix denotes the square steel tube in Polish

Number	Name of the element	Section
1	Main girder	RK 300×300×12
2	Cross-girder	RK 150×150×12
3	Bracing	RK 60×60×5
4	Arch	RK 350×350×10
5	Column	RK 60×60×4
6	Bracing	RK 70×70×6

In Tab. 2 a list of elements used in analyzed structure was presented, divided by types of cross-section and total weight. More than 75% of the supporting footbridge constructions total weight comprised of elements responsible for significant internal forces redistribution, such us girders or arc.

Tab.2: List of footbridge elements with their overall mass

Type	Number	Length (m)	Unit weight (N/m)	Bar weight (N)	Total weight (N)
S 355					
RK 60x60x4	18	52,44	67,69	3549,76	3550
RK 60x60x5	20	37,00	82,57	3055,16	3055
RK 70x70x6	16	64,08	115,72	7415,24	7415
RK 150x150x12	21	29,40	498,18	14646,43	14646
RK 300x300x8	2	48,00	714,65	34303,33	34303
RK 350x350x10	20	51,36	1039,63	53395,65	53396
Total					116366

2.4 Loads

Despite the fact that the comparative numerical analysis of obtained stress and displacement results was assumed to be held only with dead load, additional calculations concerning all possible acting loads on the footbridge should also be demonstrated. Mentioned load combinations were only used to determine the minimum required cross-section area of each footbridge element (see Tab. 1).

For the load combinations the first three groups were described as dead loads. The surface of the footbridge was made of grid deck 40×3 mm with 25.5 mm thickness [7], while the security elements – pedestrians traffic railings were made of flat bars.

Tab.3: Structural and non-structural dead loads

Number	Load type	Characteristic load value
	Structural load	
1	Footbridge elements dead load	
	Non-structural load	
2	Grid deck	1.7 kN/m^2
3	Railing	0.47 kN/m

The values of actions were taken from [8] in case of an assemblage load, whereas crowd load values were taken from [9], respectively. The values of temperature impact on steel material during different year seasons were specified on the basis of [10]. Wind load, which acted on vertical load-bearing elements of the footbridge (see items 4, 5, 6 in Tab. 1]), were calculated in accordance with the procedure set out in [11]. The first wind zone area was assumed and additional coefficient (c_f) for truss construction aerodynamic drag was taken into further considerations. Adopted values of live loads, wind and temperature were presented in Tab. 4.

Tab.4: Live loads and temperature values

Group number	Load type	Characteristic load value
	Live load	
4-7	Crowd load	4.5 kN/m^2
8-11	Assemblage load	0.8 kN/m
	Temperature	
12	Temperature summer	55^{0}C
13	Temperature winter	-20^{0}C
	Wind actions	
14	Wind from left side	0.91 kN/m
15	Wind from right side	0.91 kN/m

The influence of snow load was ignored, due to the fact that the object was designed in areas which are not exposed to permanent snow load. Additionally, designed structure had no roof.

2.5 Ultimate limit state calculations for construction elements

For the entire model, in accordance with the currently steel structures applicable standards and for the given load combinations the ultimate limit state calculations were carried out in order to obtain minimal area of cross-section for each element meeting above stated conditions. As an additional condition it was assumed that each element of structure should be utilized at its maximum capacity. Obtained results of cross-sections dimensions calculations were shown in Tab. 5.

Tab.5: Obtained results of cross-sections dimensions in Robot Structural Analysis software

Member		Section	Material	Lay	Laz	Ratio	Case
Code group : 1 Main girders							
19	OK	RK 300x300x8	S 355	201.78	201.78	0.85	16 SGN /4280/
Code group : 2 Cross-girders							
43 Pręt_43	OK	RK 150x150x12	S 355	25.04	25.04	0.97	16 SGN /3195/
Code group : 3 Bracing							
157	OK	RK 60x60x5	S 355	82.62	82.62	0.83	16 SGN /3190/
Code group : 4 Arch							
159	OK	RK 350x350x10	S 355	185.18	185.18	0.71	16 SGN /3191/
Code group : 5 Column							
66 Pręt_66	OK	RK 60x60x4	S 355	176.01	176.01	0.74	16 SGN /3191/
Code group : 6 Bracing							
119 Pręt_119	OK	RK 70x70x6	S 355	134.39	134.39	0.85	16 SGN /4319/

2.6 Numerical model in Adina software

Second static numerical analysis was carried out in Adina software. Numerical model of the arch footbridge was presented in Fig. 5. Point S in Fig. 5 denotes the location (point located at the connection of column and arch), from which maximum values of stress were read-out, whereas point D, located on the bottom surface of main girder denotes location of Z-axis displacements read-out, respectively. Adopted Boundary conditions, material properties and geometry was exactly the same as in the analyzed model in Autodesk Robot. Numerical model was created from "Body Sheet" elements located in three-dimensional space. Discretization of presented model was performed with the use of "Shell" quad (4-node) elements. The size of mesh division in the model varies from 0.10 up to 0.02m, where the highest mesh density was used near the element connections.

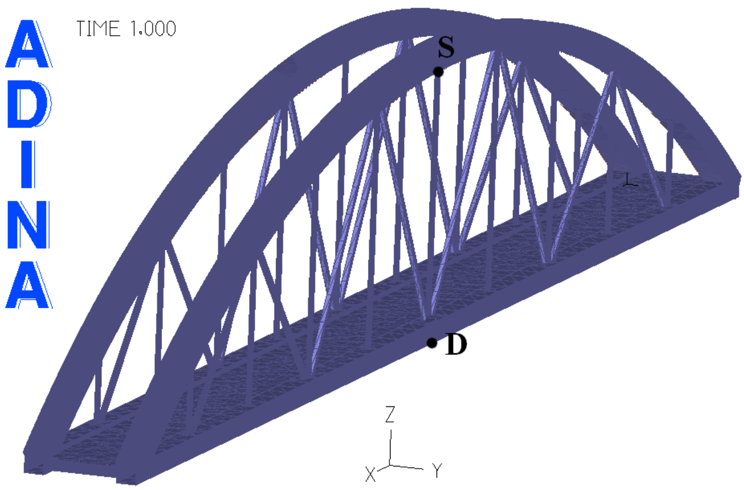


Fig.5: Numerical model prepared in the Adina software.

3 RESULTS AND DISCUSSION

The comparative analysis of stresses and displacements obtained in both FEM software i.e. ADINA and Robot Structural Analysis was performed only for the dead load acting in both models.

List of the maximum stress values in whole construction for both computational models was shown in Tab. 6.

Tab.6: Maximum stress values at point S under the dead load in both ADINA and Autodesk Robot

Type of stress	**Autodesk Robot [MPa]**	**ADINA [MPa]**
σ_{xx}	5.45	5.73
σ_{yy}	3.17	3.45
σ_{zz}	5.46	5.62
τ_{xy}	0.77	0.81
τ_{xz}	0.90	0.96
$\sigma_{effective}$ (von Misses)	6.51	6.56

In Fig. 6 effective stress redistribution band plot from ADINA program was presented. The largest concentration of stress was observed in the arc - obtained values of compressive stress were more than three times higher than in the rest of the structure.

According to the book definition, appropriately designed arc element works mainly in compression, thus it is possible to get longer spans of the bridges/footbridges then for the beam bridge constructions. Girder beams should work mainly in bending stresses.

The concentration of stress could be observed mainly near the bars connections. It should be remembered that the bearing capacity of mentioned connections should always be checked in view of maximum stress values in order to prevent further construction failure.

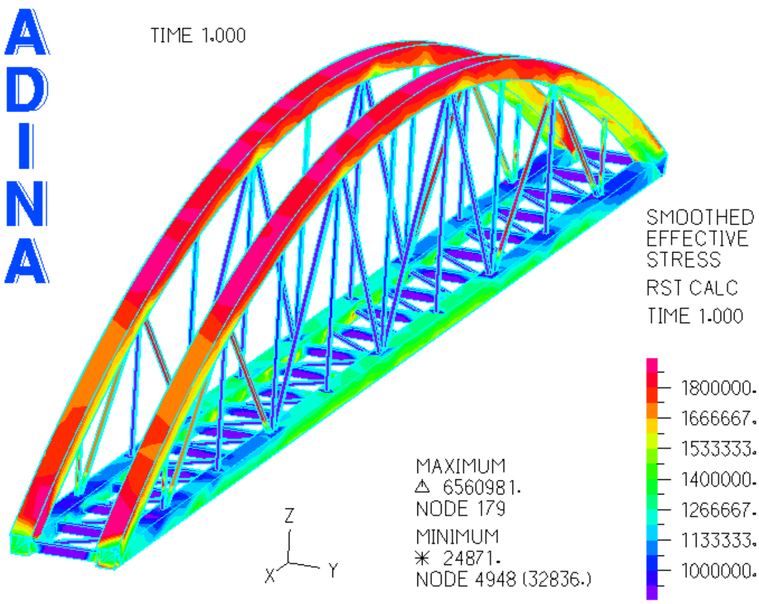
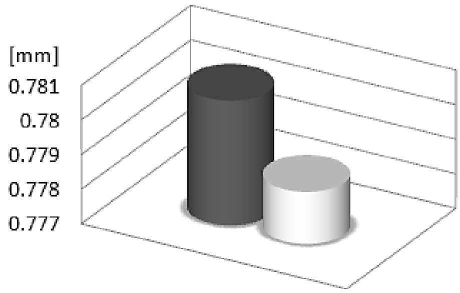


Fig.6: Effective stress plot for the dead load. Adina software

Comparison of obtained results of the Z-axis displacement in ADINA and Autodesk Robot numerical was shown in Fig. 8, whereas whole model deformation with Z-axis displacement plot was presented in Fig. 9. The maximum values of the displacement were measured in the mid-span girder beam (see Fig. 5 point D).



	Z-axis displacement [mm]
ROBOT	0.7806
ADINA	0.7787

Fig.8: Comparison of the maximum Z-axis displacement value in both numerical models

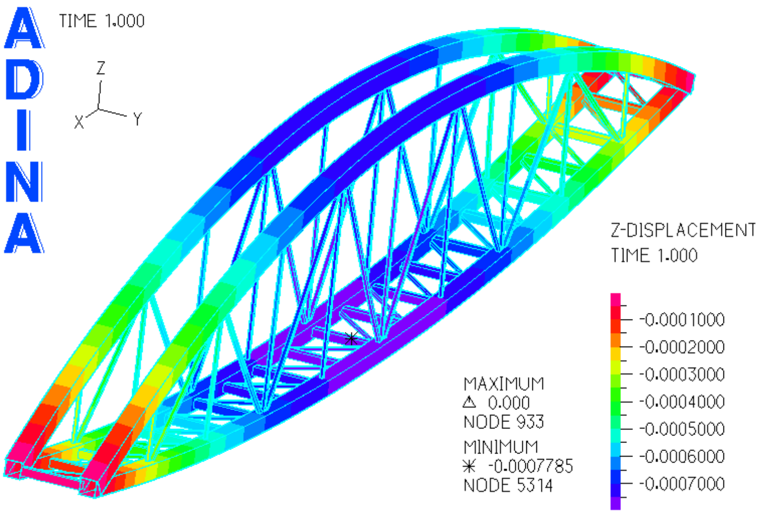


Fig.9: Z-axis displacement band plot under dead load in ADINA program

4 CONCLUSIONS

Differences in obtained results from both FEM programs were connected mainly with the simplifications used during the object modeling process. The geometry of the individual elements in ADINA software comprised of square tube elements without rounded edges in comparison with a normal square tube shape defined in Robot software. Mentioned differences were also connected with the type of used elements. Autodesk Robot model was created with "beam" elements, whereas in ADINA program it was necessary to use "Shell" type elements due to the large dimensions of the structure in relation to the small cross-section dimensions. In the ROBOT program bars had virtually added cross-section, which described the profile with rounds, whereas in ADINA, square tube rounds were omitted due to the model simplification. It is worth noting that the smaller finite elements and more nodes describes singular element, the higher total value of nodes and elements are obtained, which as a consequence requires significantly more computer memory and processor cores. In some cases, due to limited computer available memory and CPU speed it is wiser to analyze only small part of construction with adequate boundary conditions imposed.

To sum up, Autodesk Robot Structural Analysis is a program specially designed for the purpose of engineering (see also [5]). It allows create calculation model from the beginning, starting from construction geometry, applying loads, creating combinations between them, ending with the computations and cross-sections dimensions' calculation of construction element according to the national standards. In the case of ADINA program there is no possibility to make cross-section dimensions' calculation in accordance with national standards. Also creating the model geometry is far more complex than in Robot software [12]. It should be noted that despite the fact that ADINA does not have some modules, which are included in Autodesk Robot, problems of almost any area of science may be analyzed with the FEM utilization.

Selection of software to perform assumed analysis of the civil engineering structure depends mainly on the size/complexity of the object and expected form of obtained results. If it is known that the problem is little and complex, and the conditions for the Ultimate Limit State are met, but more accurate calculations are required - ADINA program should have been chosen, whereas for analyzes of large/complex structures the utilization of Autodesk Robot Structural analysis should have been considered.

LITERATURE

[1] BUJNÁK, J., HLINKA R. ODROBINÁK J. & VIČAN J. Diagnostics and evaluation of footbridges, *Procedia Engineering*, Volume 40, Pages 56-61, doi:10.1016/j.proeng.2012.07.055, 2012.

[2] VIČAN, J., GOCAL J., ODROBINAK J., MORAVCIK M. & KOTEŠ P. Determination of Railway Bridges Loading Capacity, *Procedia Engineering*, Volume 111, Pages 839-844, doi:10.1016/j.proeng.2015.07.155, 2015.

[3] KREJSA, M. & TOMICA V. Determination of inspections of structures subject to fatigue, *Transactions of the VŠB – Technical University of Ostrava, Civil Engineering Series.* Volume 11, Issue 1, Pages 1–9, ISSN (Online) 1804-4824, ISSN (Print) 1213-1962, DOI: https://doi.org/10.2478/v10160-011-0007-x, June 2011.

[4] BARCIK, W., BILISZCZUK J. & MACHELSKI C. *Projektowanie stalowych kładek dla pieszych*, Dolnośląskie Wydawnictwo Edukacyjne, Katowice 2007, ISBN: 9788371251597.

[5] MAJOR, M. & KULIŃSKI K. Comparative numerical analysis of advertising board tower using ADINA and Autodesk ROBOT Structural Analysis, *Transactions of the VŠB – Technical University of Ostrava, Civil Engineering Series*. Volume 15, Issue 2, ISSN (Online) 1804-4824, DOI: https://doi.org/10.1515/tvsb-2015-0015, January 2016.

[6] 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, Pages 79–84, ISSN (Online) 1804-4824, DOI: https://doi.org/10.2478/tvsb-2014-0023, December 2014

[7] Polimex Mostostal catalogue - grid decks (in Polish).

[8] PN-EN 1991-1-6:2007/NA:2010, Eurocode 1: Actions on structures Part 1-6: General actions - Actions during execution, PKN, Warszawa 2007. (In Polish).

[9] PN-EN 1991-2:2007, Eurocode 1: Actions on structures - Part 2: Traffic loads on Bridges, PKN, Warszawa, 2007. (In Polish).

[10] PN-EN 1991-1-5:2005/NA:2010, Eurocode 1: Actions on structures - Part 1-5: General actions – Thermal actions, PKN, Warszawa, 2005. (In Polish).

[11] PN-EN 1991-1-4:2008, Eurocode 1: Actions on structures, Part 1-4: General actions – wind actions, PKN, Warszawa, 2008. (In Polish).

[12] MAJOR, M. & MAJOR I. Comparative analysis of stress in hyperelastic Mooney-Rivlin and Zahorski materials using ADINA software, *Transactions of the VŠB – Technical University of Ostrava, Civil Engineering Series*. Volume 15, Issue 2, ISSN (Online) 1804-4824, DOI: https://doi.org/10.1515/tvsb-2015-0026, January 2016.