MACHINE DESIGN, VOL.11(2019) NO.3, ISSN 1821-1259 PP. 75-84

DOI: 10.24867/MD.11.2019.3.75-84
Preliminary note

A COMPARATIVE STUDY OF ANALYTICAL AND NUMERICAL ANALYSES OF THE STRUCTURE OF THE DOUBLE-BEAM BRIDGE CRANE WITH THE RAIL IN THE MIDDLE OF THE MAIN GIRDER

Goran PAVLOVIĆ[1, *] - Mile SAVKOVIĆ[2] - Nebojša ZDRAVKOVIĆ[2]
[1] R&D Center Alfatec Ltd, Niš, Serbia
[2] University of Kragujevac, Faculty of Mechanical and Structural Engineering, Kraljevo, Serbia

Received (02.05.2019); ***Revised*** (10.09.2019); ***Accepted*** (12.09.2019)

Abstract: *This research deals with the analysis and comparison of the results obtained analytically and by using the finite element method on the box girder of a double-beam bridge crane with the rail in the middle of the girder, where an existing bridge crane, which is in use, was observed. Stress states in the girder were observed in this analysis, and the emphasis was placed on the local stress states which occur in the middle of the girder due to the wheel-rail pressure. The expressions used in the domestic literature and applied in the design of these types of structures were used for obtaining analytical values of the results. The software package SAP2000 and the software module Autodesk Nastran In-CAD of the software package Autodesk Inventor were used for the FME analysis, where the girder structure was modelled by shell finite elements in both cases. In the application of the software package SAP2000, the rail was modelled by beam finite elements, and the loads were applied on the nodes of the finite element mesh which connect beam and plate finite elements, while in the application of the software module Autodesk Nastran In-CAD the rail was modelled by 3D finite elements, and the loads were applied on the external rail surfaces which are in contact with the wheel.*
The aim of this research is to present the comparison between analytical and numerical results in order to justify the application of the proposed models of FEM analysis, their advantages and disadvantages, as well as the credibility of the obtained results in relation to the existing analytical model of calculation. In this way, it can be shown whether the FEM results are within those obtained analytically, so that in the design of these types of structures the proposed modelling methods could be reliably used.

Key words: *Box girder, Bridge crane, Local stresses, SAP2000, Autodesk Nastran In-CAD*

1. INTRODUCTION

Double-beam bridge cranes are most widely spread and have the largest application in relation to the other types of cranes, which are used in industrial halls and plants.

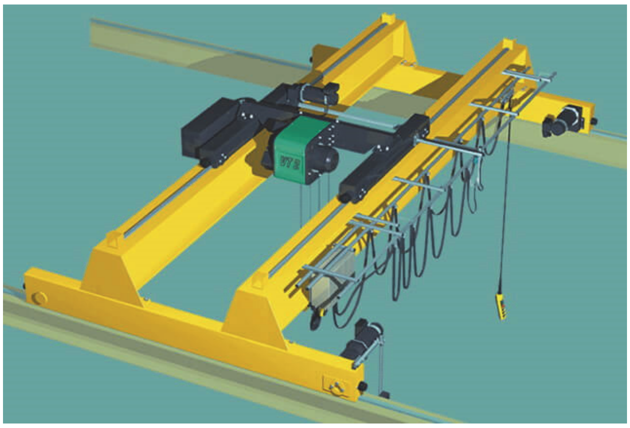

Fig.1. Double-beam bridge crane with the rail in the middle of the main girder

They are most often applied in load manipulation in production as well as for maintenance in various types of plants.

Nowadays, judging by the practice of the leading manufacturers of crane equipment, double-beam bridge cranes with the rail above the web are increasingly used but, regardless of this fact, these structures with the rail in the middle of the main girder are still present (Figure 1).
Also, there are numerous publications which deal with the problem of analysis and optimization of double-beam bridge cranes with the rail in the middle of the main girder, as well as with their comparison with the mentioned type of solution. In addition to stresses and strains, analyses increasingly include the other important criteria.
The analysis and optimization of the girder in a double-beam bridge crane with the rail in the middle of the main girder is most often performed by applying FEM, i.e. by using 3D finite elements, [1-10, 12, 13], indicating the importance and justification of application of the given software packages.
The paper [1] presents the design of main girders in a double-beam bridge crane, with the application of the CAE platform based on ANSYS software. It shows a considerable saving in time by using this type of design, where maximum stresses, strains and fatigue are taken into account. ANSYS is largely applied in these types of structures, which can also be seen in [2], which presents the analysis of an existing bridge crane designed

*Correspondence Author's Address: R&D Center Alfatec Ltd, Bulevar Nikole Tesle 63/5, 18000 Niš, Serbia, goran.pavlovic@alfatec.rs

according to the Indian standard, according to stresses and deflections, where the crane model was created in Creo software. The bridge crane designed according to the Indian standard is also analyzed in [3], where the comparison between analytical and numerical results was carried out, and the model of the carrying structure of the bridge crane was obtained in Pro/E software. Similarly to the previous research, the comparison between the analytical method and the FEM results was performed, with the stresses reduced by shaping sharp edges, [4]. Also, ANSYS software was applied for the optimization of the mass of the main girders in a double-beam bridge crane, [5], where the saving of 19.4 % was achieved, taking into account stresses and deflections of the structure. Similarly, the paper [6] presents the optimization of the mass of the main girders of an existing bridge crane and thus the realized saving was 29 %.

The paper [7] deals with the analysis and optimization of the whole carrying structure of a double-beam bridge crane, observing the connection between the end girders and the main girders and its influence on the decrease of stresses in the structure, while the paper [8] provides the analysis of a double-beam bridge crane aiming at presenting the critical zones of stresses and deflections, by using Radioss 12.0 and HyperWorks software packages. Also by applying HyperWorks software, the research presented in [9] dealt with the multicriteria optimization of the main girders of a bridge crane, with the optimizations done by shape, size and topology as well as with the oprimization of the cross section, which resulted in a significant saving in the mass. Varying of plate thickness and positions of vertical stiffeners was carried out in [10], where optimization of the existing main girders of the bridge crane, with the mass reduced by as much as 38%, was done according to the criteria of stresses, deflections and natural frequencies. SolidWorks software was used here for the model of the bridge crane structure.

In contrast to the previous research, where 3D finite elements were used, shell finite elements were used in the paper [11], in which the model of a double-beam bridge crane was prepared in FEMAP software, while the static and dynamic analyses were performed by using PAK software.

In the research presented in [12], the model of a bridge crane with the rail in the middle of the main girder was compared with two modern variants of the solution with the rail above the web, based on maximum stresses, by applying 3D finite elements, where a slight saving accomplished by new variants of solution of main girders was shown. Similarly to the previous research, in the master's thesis [13], the author presented the analysis of the main girder of a double-beam bridge crane with the rail in the middle of the girder, both analytically and through FEM, and compared it with the variants of the girder with the rail above the web, where the advantages of these solutions over the existing ones was shown.

Unlike FEM, analysis and optimization can be performed in a pure analytical way, which is shown in papers [14-16]. The paper [14] compares two types of bridge cranes analytically, and a concrete example is used to show that the main girders have a smaller mass if the rail is above the web if compared to those with the rail in the middle of the main girder. The research described in paper [15] dealt with the problem of optimization of main girders with the rail above the web, with the emphasis placed on maximum stresses, i.e. the occurrence of biaxial stress state, and the application of the given model was justified in two examples of cranes, by using HS and SA optimization algorithms. Application of the well-known analytical optimization process, the method of Lagrange multipliers, was shown in [16], where an example of a bridge crane was used to show how the optimum geometrical parameters of a box girder change depending on the selection of materials and conditions in which the crane opertaes, based on the stress states occuring in the upper flange of the girder.

The presented publications point to the importance and justification of considerations of this type of engineering problems as well as to the application of most diverse software packages and analysis procedures.

The aim of this research is to use an example of a solution of double-beam bridge crane with the rail in the middle of the girder to present the comparison between analytical and numerical results and, consequently, draw certain conclusions and guidelines for the design of this type of structures.

2. ANALYTICAL CALCULATION OF THE MAIN GIRDER

The calculation of the main girder will be performed on the basis of strength, i.e. stress states in the middle of the girder, below the rail, at the point of maximum stress, according to [17].

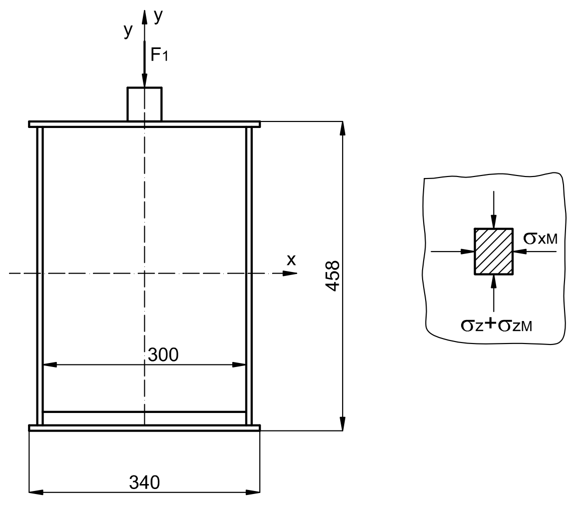

Fig.2. Cross section of the box girder with the presentation of stress states in the flange plate below the rail

A solution of double-beam bridge crane, with the carrying capacity of 10 t and the span of 7 m, will be used as an example. The diaphragms within the girders are distributed at a distance of 1 m. The thickness of all girder plates and diaphragms is 8 mm.

Figure 2 presents the cross section of the main girder of a double-beam bridge crane with the rail in the middle of the girder, with necessary dimensions, for the observed example.

2.1. Stress calculation

In the analysis of stress states in the box girder, the influence of local stress which occurs due to the pressure of the winch wheel on the rail, when additional stress components in the *z* and *x* directions occur, will be primarily observed. That is the reason why the stress state will be observed in the middle of the girder (below the rail), where two load positions will be observed when wheel 1 ($F_1 > F_2$) is in the middle between two diaphragms (since maximum values of local stresses occur at that point): position 1 is when the force F_1 is in the middle of the third zone (Figure 3), and position 2 is when F_1 is in the middle of the fourth zone, i.e. in the middle of the girder span (Figure 4).

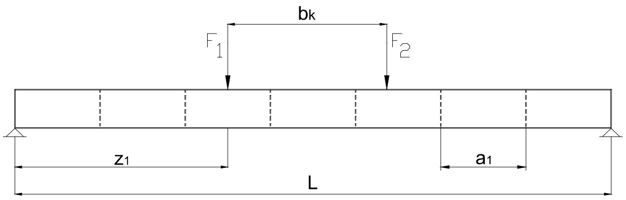

Fig.3. Static model of the girder for load position 1

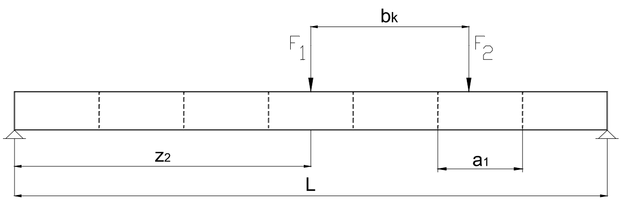

Fig.4. Static model of the girder for load position 2

The continuation of the paper will present all necessary relations which are used for the static calculation of the girder.

$$R = \gamma \cdot \frac{\psi \cdot Q + m_k}{2} \cdot g \quad (1)$$

$$F_1 = R \cdot \frac{b_k - e_1}{b_k} \quad (2)$$

$$F_2 = R - F_1 \quad (3)$$

$$M_1 = \frac{F_1 \cdot z_1 \cdot (L - z_1)}{L} + \frac{F_2 \cdot b_1 \cdot z_1}{L} + +\gamma \cdot \frac{q_n \cdot (L \cdot z_1 - z_1^2)}{2} \quad (4)$$

$$b_1 = L - (z_1 + b_k) \quad (5)$$

$$z_1 = 2 \cdot a_1 + \frac{a_1}{2} \quad (6)$$

$$M_2 = \frac{F_1 \cdot L}{4} + \frac{F_2 \cdot b_2 \cdot z_2}{L} + \gamma \cdot \frac{q_n \cdot L^2}{8} \quad (7)$$

$$b_2 = L - (z_2 + b_k) \quad (8)$$

$$z_2 = \frac{L}{2} \quad (9)$$

$$q_n = 1.1 \cdot \rho \cdot g \cdot A \quad (10)$$

where:

R - the resulting force in the main girder,

F_1, F_2 - the forces of pressure on the girder, from wheel 1 and wheel 2, respectively,

γ, ψ - the coefficients, according to [17],

M_1, M_2 - the maximum bending moments at the points z_1 and z_2, respectively,

q_n - the specific weight of the girder (increased by 10 % due to the weight of the diaphragms and the welds).

Figure 5 shows the position of wheel 1 which is located between two diaphragms, when maximum local stresses occur, as well as the influential zone of action of pressure on the girder upper plate, size z_0.

$$z_0 = 2 \cdot h_{\check{s}} + 5, \ cm \quad (11)$$

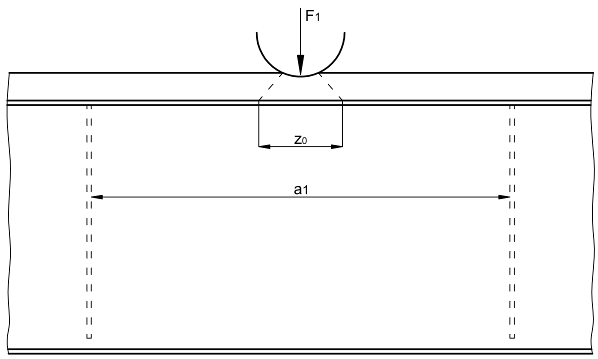

Fig.5. Position of the diaphragms in the interior of the girder

The calculation of values necessary for determination of local stresses is presented in the continuation of the text.

$$N = \frac{F_1}{1 + \frac{96 \cdot b_1^2 \cdot I_{\check{S}1} \cdot K_1}{a_1^3 \cdot t_1^3} \cdot \frac{1}{c_o}}, \quad a_1 \geq b_1 \quad (12)$$

$$N = \frac{F_1}{1 + \frac{96 \cdot I_{\check{S}1} \cdot K_1}{a_1 \cdot t_1^3} \cdot \frac{1}{c_o}}, \quad a_1 < b_1 \quad (13)$$

$$\sigma_{z,u} = \sigma_z + \sigma_{zM} \leq \sigma_d \quad (14)$$

$$\sigma_d = \frac{R_e}{\nu_1} \quad (15)$$

$$\sigma_z = \frac{M}{W_x} \quad (16)$$

$$\sigma_{zM} = \frac{6 \cdot K_3 \cdot N}{t_1^2} \tag{17}$$

$$\sigma_{xM} = \frac{6 \cdot K_2 \cdot N}{t_1^2} \le \sigma_d \tag{18}$$

$$\sigma_u = \sqrt{\sigma_{z,u}^2 + \sigma_{xM}^2 - \sigma_{z,u} \cdot \sigma_{xM}} \le 1.1 \cdot \sigma_d \tag{19}$$

where:
N - part of the wheel pressure force which, due to the rail stiffness, is transferred to the plate,
a_1 - the distance between the diaphragms,
b_1 - the inner width between the webs of the box girder (300 mm, Figure 2),
t_1 - the upper flange plate thickness,
K_1, K_2, K_3, c_o - the coefficients, according to [17],
$I_{Š1}$ - the moment of inertia of the rail for its axis,
σ_{zM}, σ_{xM} - the stress components in the longitudinal and cross directions, due to local bending, respectively,
σ_z, $\sigma_{z,u}$ - the normal stress due to bending and the total stress in the longitudinal direction, respectively,
σ_d - the permissible stress of the girder material,
R_e - the yield stress for the basic material of the main girder,
$\nu_1 = 1.5$ - the permissible stress of the girder material,
σ_u - the equivalent (total) stress in the middle of the girder upper plate.
The maximum stress in the rail is determined based on the following relation:

$$\sigma_š = \frac{N_1 \cdot a_1}{6 \cdot W_{x,š}} \le \sigma_{š,d} \tag{20}$$

$$N_1 = F_1 - N \tag{21}$$

$$\sigma_{š,d} = \frac{R_{e,š}}{1.3} \tag{22}$$

where:
N_1 - part of the wheel pressure force which is connected with the rail,
$W_{x,š}$ - the resistance moment of inertia of the rail,
$\sigma_{š,d}$ - the permissible stress of the girder material,
$R_{e,š}$ - the yield stress for the rail material.
The input data (characteristics of the existing crane) necessary for the calculation of the girder are as follows:

- the carrying capacity of the crane: Q = 10 t,
- the span of the crane: L = 7 m,
- the weight of the trolley with the winch: m_k = 4.49 t,
- the span of the trolley wheels: b_k = 1866 mm,
- the position of the resultant force of load in relation to the wheel 1 - trolley: e_1 = 783 mm,
- the material of the girder plates S235 (R_e = 23.5 kN/cm^2),
- the classification class of the crane: 2,
- the rail dimensions: 50x50,
- the rail material: S355 (R_e = 35.5 kN/cm^2).

The other data were taken according to the classification class, according to [17].

2.2. Analytical results

Based on the input parameters, the calculation results in obtaining the value of stress in the rail $\sigma_š$ = 23.68 kN/cm^2, while the permissible stress of the rail material is $\sigma_{š,d}$ = 27.31 kN/cm^2.
Table 1 shows the values of stress components in the middle of the girder upper plate as well as the equivalent stress at the point of pressure of wheel 1 on the rail.

Table 1. Analytical values of stresses in the box girder

Case	σ_z (kN/cm^2)	σ_{zM} (kN/cm^2)	$\sigma_{z,u}$ (kN/cm^2)	σ_{xM} (kN/cm^2)	σ_u (kN/cm^2)
1.	3.78	23.21	26.99	35.49	32.10
2.	3.89		27.09		32.13

As it can be seen from Table 1, the normal bending stress σ_z has a relatively small value in relation to the local stresses σ_{zM} and σ_{xM}, and therefore the emphasis is placed on the middle of the girder, on the stress below the rail.
During the design phase and calculation of these girders, the stress states of local stresses are not taken into account, so that based on Table 1 it can be seen that the exceeding of stress occurs in these components (σ_d = 15.67 kN/cm^2) and also that the total stress is exceeded ($1.1 \cdot \sigma_d$ = 17.23 kN/cm^2).
If a short diaphragm were inserted between the existing diaphragms, in each zone, these stresses would considerably decrease, so that their values would be σ_{zM} = 4.74 kN/cm^2, σ_{xM} = 6.82 kN/cm^2. In this case, the stress states would not be exceeded.

3. NUMERICAL ANALYSIS OF THE MAIN GIRDER

3.1. Results from SAP2000

Figure 6 presents a hybrid model for the FEM analysis in SAP2000 software, which consists of a main girder modelled by shell finite elements and a rail modelled by beam finite elements.

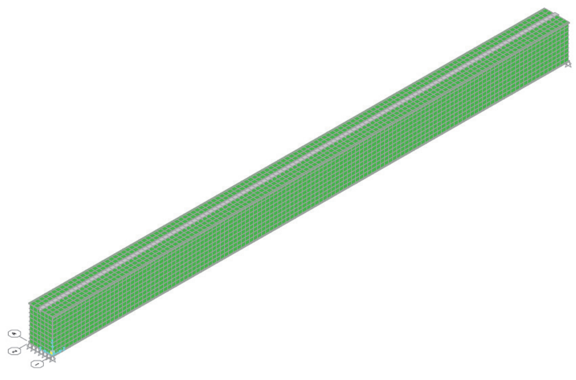

Fig.6. Mesh of finite elements of the girder with the rail in SAP2000

The following figures presents the diagrams of distribution of bending moments and shear force in the rail at the point of action of wheel 1 (point with the largest load) for both observed load positions (Figure 7 and Figure 8).

Fig.7. Static diagrams at the point of maximum load of the rail for load position 1

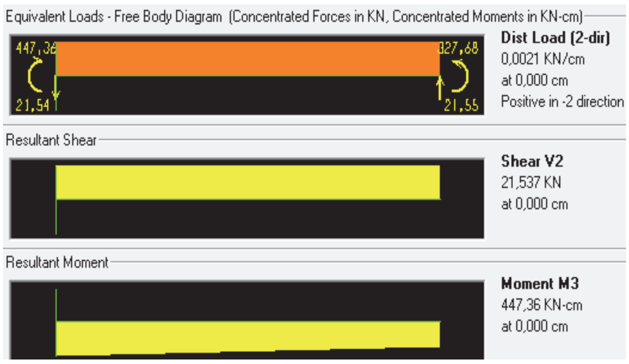

Fig.8. Static diagrams at the point of maximum load of the rail for load position 2

The following figures present the distributions of stress components in the main girder for both observed load positions (Figure 9 – Figure 14).

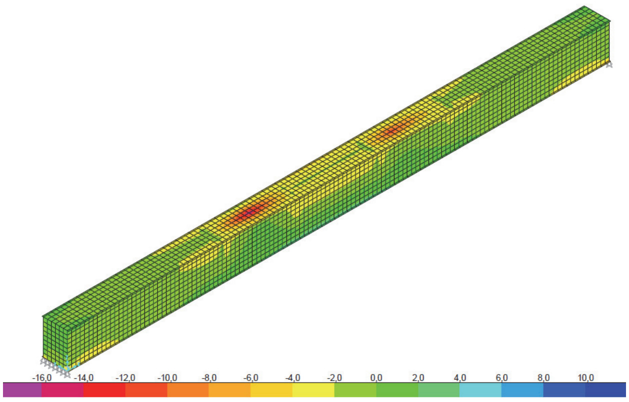

Fig.9. Distribution of stress in the z-direction in the main girder for load position 1 in SAP2000

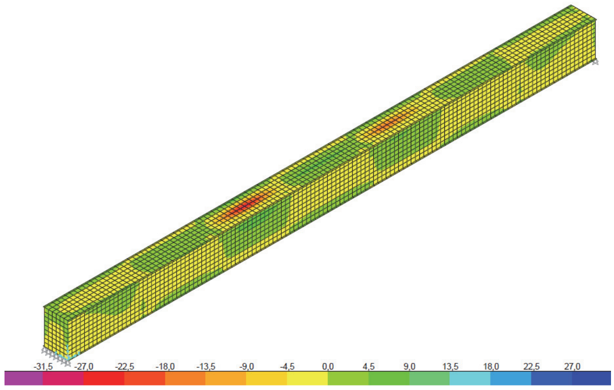

Fig.10. Distribution of stress in the x-direction in the main girder for load position 1 in SAP2000

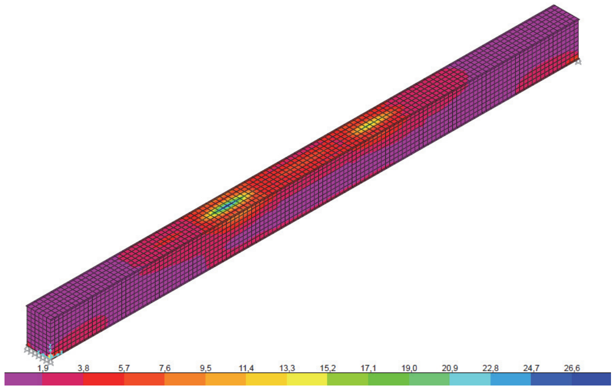

Fig.11. Distribution of the total stress in the main girder for load position 1 in SAP2000

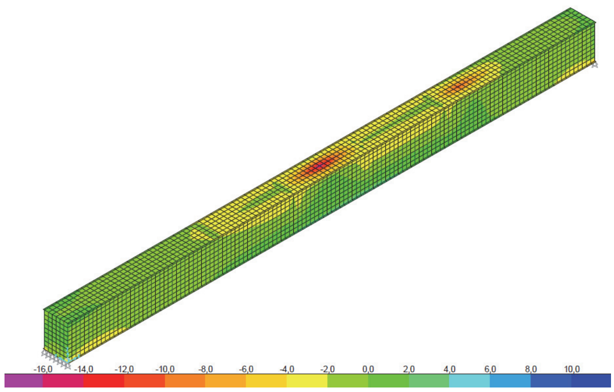

Fig.12. Distribution of stress in the z-direction in the main girder for load position 2 in SAP2000

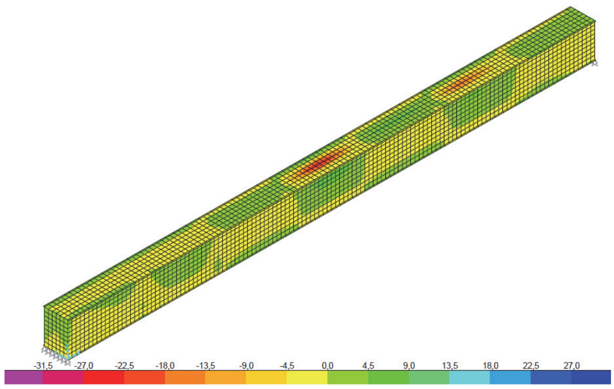

Fig.13. Distribution of stress in the x-direction in the main girder for load position 2 in SAP2000

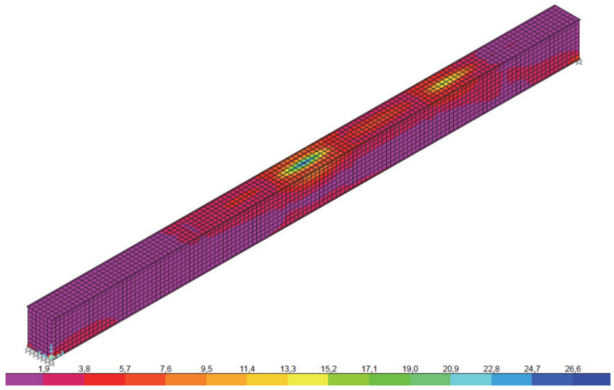

Fig.14. Distribution of the total stress in the main girder for load position 2 in SAP2000

Table 2 presents the maximum values of stress components in the main girder for both load positions, equivalent (total) stress, as well as the calculated value of stresses in the rail based on the maximum bending moment and shear force ($M_{max,1}$ = *446.81 kNcm*, $F_{s,1}$ = *21.52 kN* – for load position 1 and $M_{max,2}$ = *447.36 kNcm*, $F_{s,2}$ = *21.54 kN* – for load position 2).

Table 2. Values of stress in the box girder and the rail in SAP2000

Case	$\sigma_{z,u}$ (kN/cm^2)	σ_{xM} (kN/cm^2)	σ_u (kN/cm^2)	$\sigma_š$ (kN/cm^2)
1.	17.62	31.65	27.47	21.50
2.	17.78	31.65	27.48	21.52

3.2. Results from Autodesk Nastran In-CAD

Figure 15 presents a hybrid model for the FEM analysis in the software module Autodesk Nastran In-CAD, which consists of a main girder modelled by shell finite elements and a rail modelled by 3D finite elements.

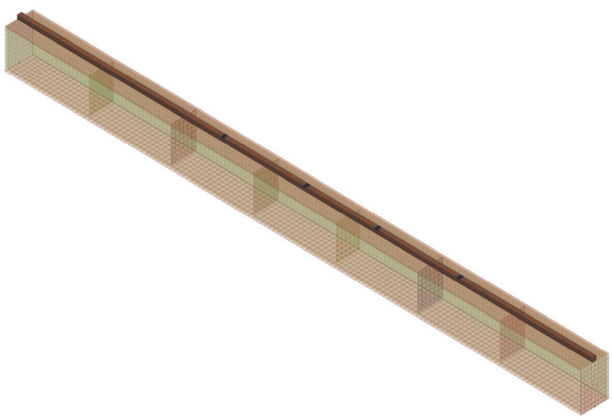

Fig.15. Mesh of finite elements of the girder with the rail in Autodesk Nastran In-CAD

The following figures present the distributions of stress components in the main girder for both observed load positions (Figure 16 – Figure 31).

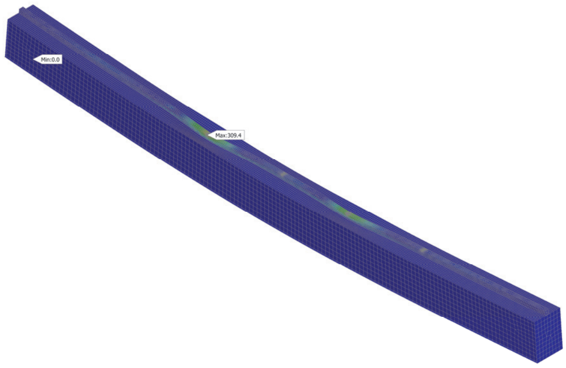

Fig.16. Maximum stress in the rail for load position 1 in Autodesk Nastran In-CAD

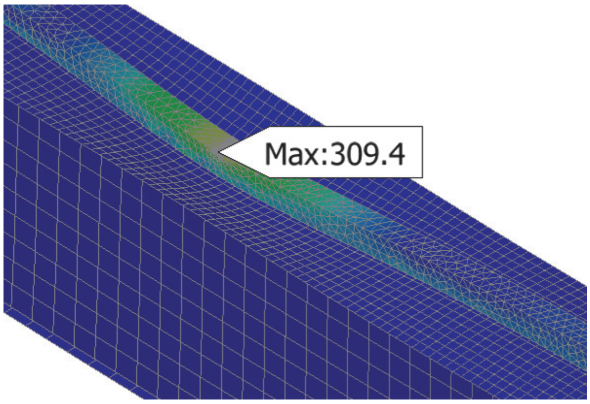

Fig.17. Detail of the presentation of the maximum stress in the rail for load position 1 in Autodesk Nastran In-CAD

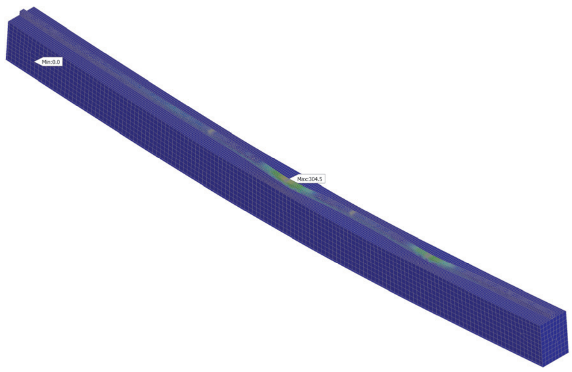

Fig.18. Maximum stress in the rail for load position 2 in Autodesk Nastran In-CAD

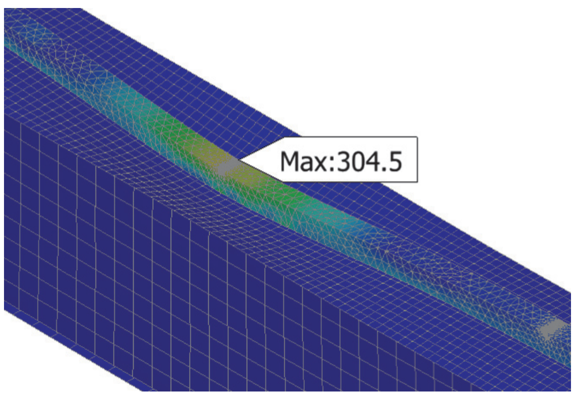

Fig.19. Detail of the presentation of the maximum stress in the rail for load position 2 in Autodesk Nastran In-CAD

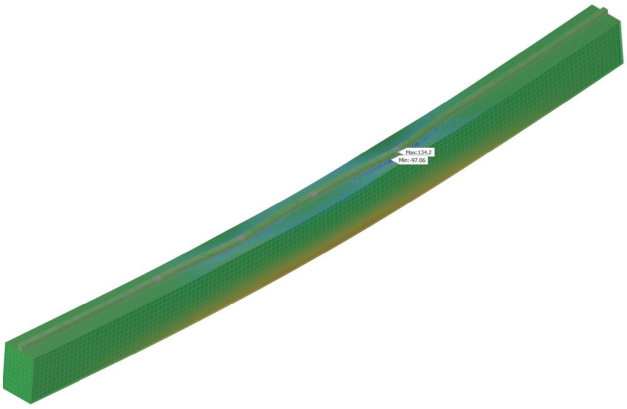

Fig.20. Distribution of stress in the z-direction in the main girder for load position 1 in Autodesk Nastran In-CAD

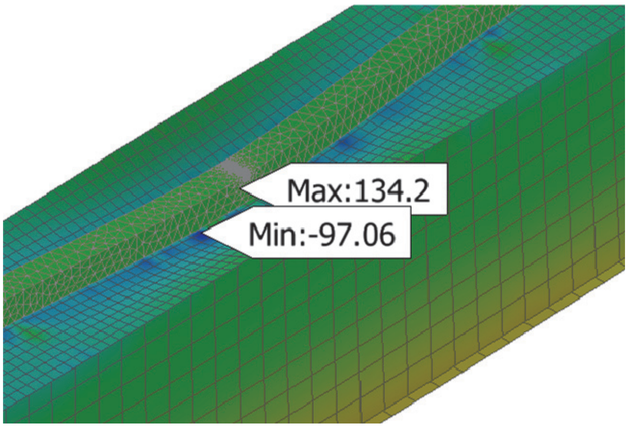

Fig.21. Detail of the distribution of stress in the z-direction in the main girder for load position 1, at the point of action of wheel 1 in Autodesk Nastran In-CAD

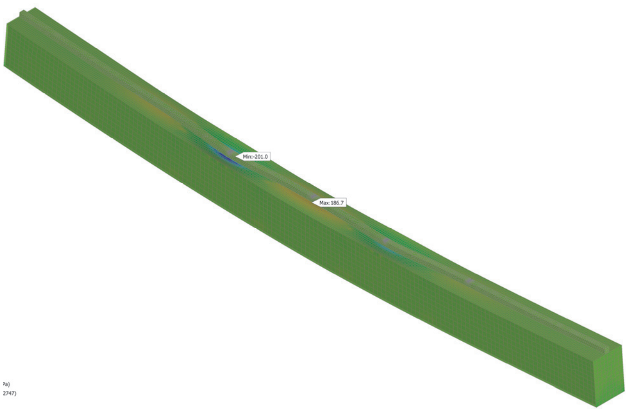

Fig.22. Distribution of stress in the x-direction in the main girder for load position 1 in Autodesk Nastran In-CAD

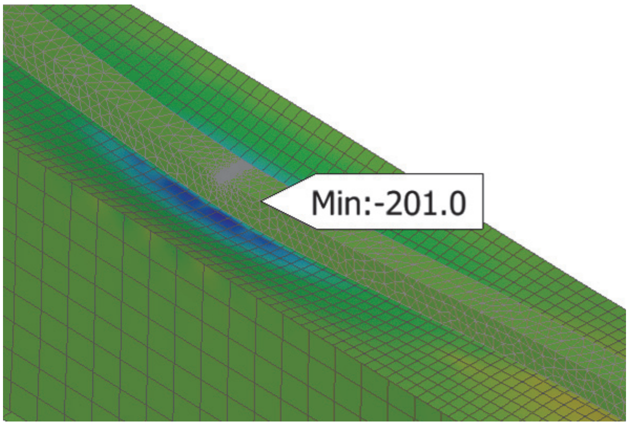

Fig.23. Detail of the distribution of stress in the x-direction in the main girder for load position 1, at the point of action of wheel 1 in Autodesk Nastran In-CAD

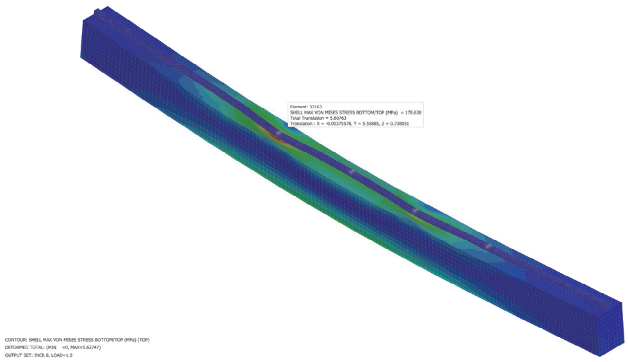

Fig.24. Distribution of the total stress in the main girder for load position 1 in Autodesk Nastran In-CAD

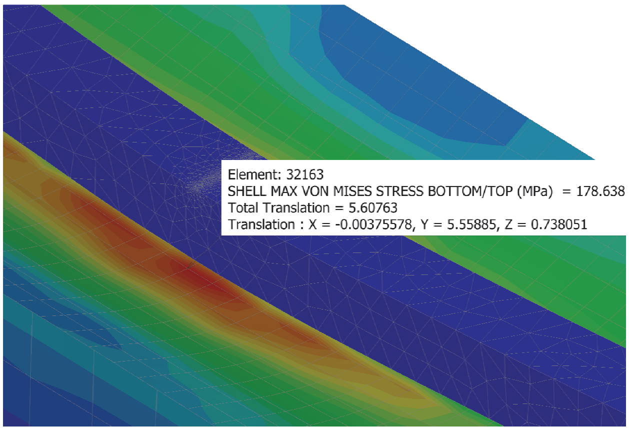

Fig.25. Detail of the distribution of the total stress in the main girder for load position 1, at the point of action of wheel 1 in Autodesk Nastran In-CAD

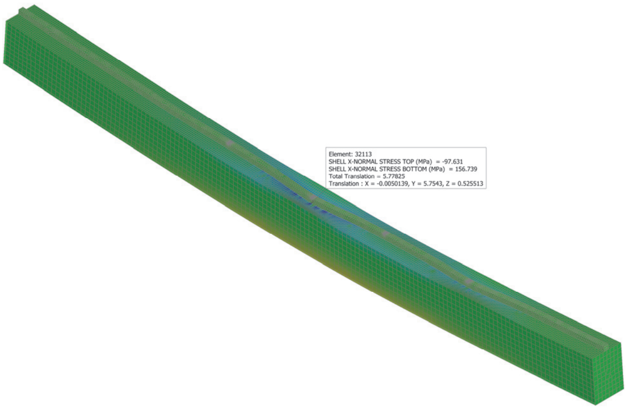

Fig.26. Distribution of stress in the z-direction in the main girder for load position 2 in Autodesk Nastran In-CAD

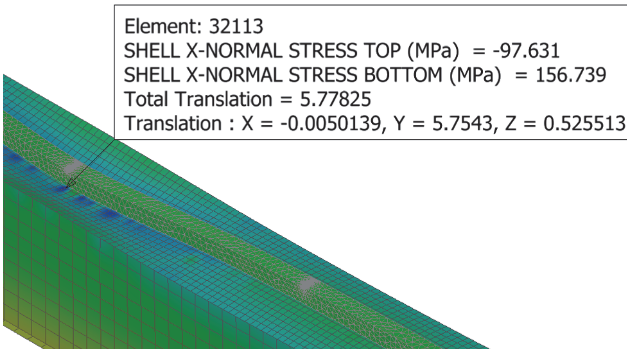

Fig.27. Detail of the distribution of stress in the z-direction in the main girder for load position 2, at the point of action of wheel 1 in Autodesk Nastran In-CAD

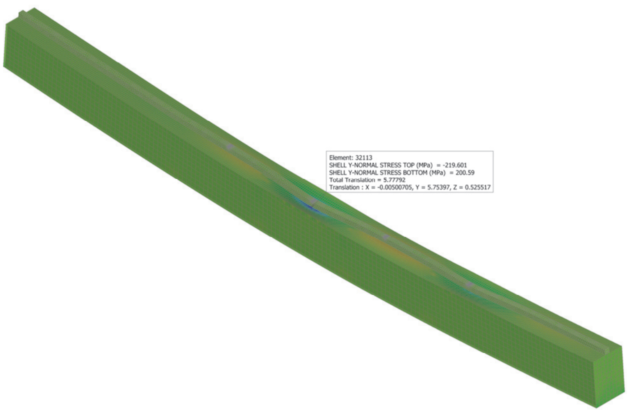

Fig.28. Distribution of stress in the x-direction in the main girder for load position 2 in Autodesk Nastran In-CAD

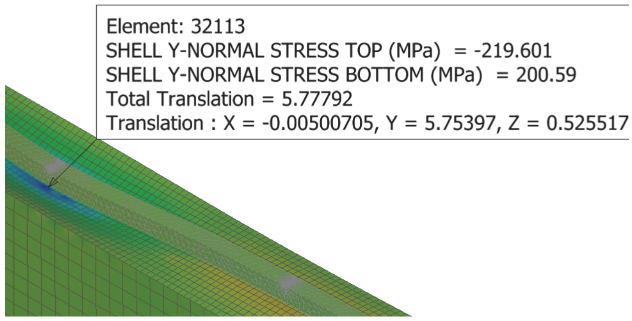

Fig.29. Detail of the distribution of stress in the x-direction in the main girder for load position 2, at the point of action of wheel 1 in Autodesk Nastran In-CAD

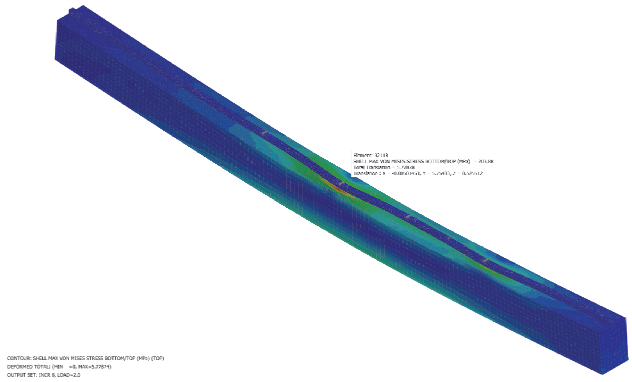

Fig.30. Distribution of the total stress in the main girder for load position 2 in Autodesk Nastran In-CAD

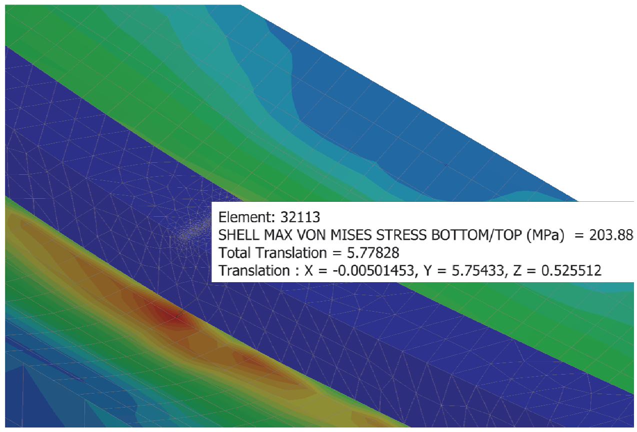

Fig.31. Detail of the distribution of the total stress in the main girder for load position 2, at the point of action of wheel 1 in Autodesk Nastran In-CAD

Table 3 presents the maximum values of stress components in the main girder for both load positions, equivalent (total) stress and the value of stress in the rail.

Table 3. Values of stresses in the box girder and the rail in Autodesk Nastran In-CAD

Case	$\sigma_{z,u}$ (kN/cm²)	σ_{xM} (kN/cm²)	σ_u (kN/cm²)	$\sigma_š$ (kN/cm²)
1.	13.42	20.10	17.86	30.94
2.	15.67	21.96	20.39	30.45

4. CONCLUSION

This research led to the comparison between analytical results obtained on the basis of [17], for the calculation of stress states in the middle of the main girder with box cross-section, as well the results obtained by FEM, in two different software packages, according to different models.

Based on Table 1 and Table 2, it is seen that the model in SAP2000 software shows quite good results regarding the stress state of the box girder in the *x*-direction (cross direction), while the value of stresses in the *z*-direction (longitudinal direction) is considerably smaller than the analytical value. Also, the value of stresses in the rail obtained in this way is approximate to the analytical value. The application of the method of modelling in this software package is thus justified and the analytical model is confirmed regarding the components of wheel pressure force and its distribution on the rail and the box girder upper plate.

Based on Table 1 and Table 3, it is seen that the stress components in the box girder in both directions are considerably lower than the analytical values, and based on Table 2 and Table 3, it is seen that the stresses in the upper plate are lower than those obtained according to the previous model. The stresses in the rail are considerably higher than those obtained analytically and the stresses obtained in the previous model. The cause can be found, before all, in the method of modelling the rail itself and its connection with the box girder, as well as in the introduction of the load itself, along the upper surface of rail.

In this case, SAP2000 provides more precise results, primarily because the modelling method corresponds more to the analytical model of calculation. The software module Autodesk Nastran In-CAD is simpler and more practical in engineering work regarding design and analysis of these types of structures. Possible modifications of the model are more easily performed in Autodesk Inventor software, which also holds for a new analysis in this module, which is not the case with the application of SAP2000, where modifications are much more complicated.

It can be noticed that the stresses in the plate obtained by FEM are within the limits in both software packages if compared to the results obtained analytically, which means that these procedures can be successfully used as reliable ones in the design of these types of carrying structures.

Also, it can be noticed that proper placement of additional short stiffeners between diaphragms can considerably decrease the value of local stresses in the girder upper plate.

Future research can deal with the analysis of local stability of box girder plates, both the upper flange plate and the webs, as well as with the influence of placing horizontal stiffeners on the webs. The inclusion of other criteria and technological restrictions is desirable for the purpose of optimization of all geometrical parameters of the box girder. The other types of models in FEM analysis, the influence of selection of the type of finite elements and the method of load introduction can also be used in order to obtain as realistic model as possible in the analysis and design of such types of structures.

ACKNOWLEDGMENT

A part of this research is a contribution to the projects of the Ministry of Education, Science and Technological Development of the Republic of Serbia, Project III 44006 and Project TR 35038.

REFERENCES

[1] Zhang, Q.; Pei, C. & Guo, J. (2012). Development of the Traveling Crane Main Girder CAE Platform Based on ANSYS, *Proceedings of 2012 International Conference on Measurement, Information and Control (MIC 2012)*, May 18-20, Harbin, China, ISBN: 9781457716010, pp. 1067-1071, Institute of Electrical and Electronics Engineers (IEEE), New York City

[2] Paresh, P. & Kamdar, N. (2014). Design and Analysis of Major Component of 120 Tones Capacity of EOT Crane. *IJEDR,* Vol. 2, No. 2, pp. 2150-2154, ISSN: 2321-9939

[3] Khalidurfeasif, P. & Bharti, D. (2015). Finite Element Analysis Of 450T EOT Crane Box Girder. *IJAERD,* Vol. 2, No. 11, pp. 171-178, ISSN: 2348-6406

[4] Sushma, K. & Malviya, A. (2017). Study & Design Analysis on Girder of EOT Crane. *IJARIIE,* Vol. 3, No. 5, pp. 1441-1445, ISSN: 2395-4396

[5] Ning, Z. (2012). Structural Optimization Research on Girder of 200t Bridge Crane Based on ANSYS. *Advanced Materials Research,* Vols. 430-432, pp. 1708-1711, ISSN: 1662-8985

[6] Suratkar, A.; Shukla, V. & Zakiuddin, K. (2013). Design Optimization of Overhead EOT Crane Box Girder using Finite Element Analysis. *IJERT,* Vol. 2, No. 7, pp. 720-724, ISSN: 2278-0181

[7] Sankar, V.; Pravin, A. & Suresh, P. (2018). Analysis and Design Optimisation in the Structural Member of a Double Girder Overhead Crane. *IJPAM,* Vol. 118, No. 11, pp. 211-217, ISSN: 1311-8080

[8] Linear Static Analysis of a Double Girder EOT Crane Assembly http://www.gowra.com/resources.html - *Gowra Engineering Technologies Pvt. Ltd.*, Case Study

[9] Yifei, T.; Zhaohui, T.; Wei, Y. & Zhen, Y. (2013). Research on Energy-saving Optimization Design of Bridge Crane. *Eksploatacja i Niezawodnosc – Maintenance and Reliability,* Vol. 15, No. 4, pp. 449–457, ISSN: 1507-2711

[10] Bećirović, A.; Vukojević, D. & Hadžikadunić, F. (2011). Optimization of Double Box Girder Overhead Crane in Function of Cross Section Parameter of Main Girders, *Proceedings of XV International Research/Expert Conference Trends in the Development of Machinery and Associated Technology (TMT 2011)*, September 12-18, Prague, Czech Republic, ISBN: 1840-4944, pp. 641-644, Faculty of Mechanical Engineering of University of Zenica, Bosnia and Herzegovina

[11] Miloradović, N.; Vujanac, R. & Slavković, R. (2006). Modeling of the Steel Structure of Overhead Traveling Crane using Numerical Methods, *Proceedings of IV Symposium with International Participation Konstruisanje Oblikovanje Dizajn (KOD 2006)*, May 30-31, Palić, Serbia & Montenegro, ISBN: 86-85211-92-1, pp. 57-60, Faculty of Technical Sciences of University of Novi Sad, Serbia

[12] Sankar, V.; Vijayan, D. & Ashraf, I. (2015). Reducing the Structural Mass of a Real-World Double Girder Overhead Crane. *IJAET,* Vol. 8, No. 2, pp. 150-162, ISSN: 22311963

[13] Lerga, J. (2011). Modeling and Simulation Research on the Metal Structure of Bridge Cranes, *Technical University - Sofia*, Master thesis

[14] Kumar, A. (2012). Economic Design of EOT Cranes, *Alard College of Engineering - Pune University*, Seminar Report

[15] Pavlović, G.; Savković, M.; Marković, G. & Zdravković, N. (2018). Mass-Optimized Design of the Main Girder with Box-Section of the Double-Girder Bridge Crane with the Rail Placed in the Middle of the Top Flange, *Proceedings of IV Conference on Mechanical Engineering Technologies and Applications (COMETa 2018)*, November 27-30, East Sarajevo – Jahorina, B&H, ISBN: 978-99976-719-4-3, pp. 343-350, Faculty of Mechanical Engineering East Sarajevo, Republic of Srpska

[16] Pavlović, G.; Gašić, M.; Savković, M.; Bulatović, R. & Zdravković, N. (2014). Optimization of the Box Section of the Main Girders of the Bridge Crane for the Case of Placing the Rail in the Middle of the Top Flange, *Proceedings of VIII International Conference Heavy Machinery (HM 2014)*, June 25-28, Zlatibor, Serbia, ISBN: 978-86-82631-74-3, pp. 105-112, Faculty of Mechanical and Civil Engineering Kraljevo, Serbia

[17] Ostrić, D. & Tošić, S. (2005). *Cranes*. Institute for Mechanization of the Faculty of Mechanical Engineering of the University in Belgrade, ISBN: 86-7083-520-7, Belgrade, Serbia