WARSAW UNIVERSITY OF TECHNOLOGY Index 351733
FACULTY OF CIVIL ENGINEERING
COMMITTEE FOR CIVIL AND WATER ENGINEERING
POLISH ACADEMY OF SCIENCES ISSN 1230-2945

DOI: 10.24425/ace.2024.148905

ARCHIVES OF CIVIL ENGINEERING

Vol. LXX ISSUE 1 2024

pp. 153 –165

Research paper

Influence of the type of numerical model a prestressed concrete bridge on the determination of its internal forces and displacements

Radosław Oleszek[1], Wojciech Radomski[2], Krzysztof Nowak[3]

Abstract: Static analyses of bridge structures are currently performed using the finite element method (FEM). Depending on the geometry of the structure and the technically required accuracy of calculations, different levels of discretization of these structures are used in their design. In the design process, beam grillage models (denoted e^1, p^2), shell models (denoted e^2, p^2) or shell-beam models (denoted $e^1 + e^2$, p^3) are often used. Solid models (denoted $e^3 + p^3$) are mostly used in advanced analyses, having frequently a scientific character. It is shown that there is an impact of the applied types of the numerical model (i.e., degree of complexity, degree of discretization, accuracy of the model) of the road bridge on the calculated values of bending moments and displacements, which indirectly affects the global safety coefficient of the designed bridge structure. The main purpose of the calculations is to examine the discrepancies of analyzed internal forces and displacements depending of the type of numerical model used. The calculated values are referred to the results taken from the field tests of the existing bridge denoted MS 03, which is a continuous beam structure with the three spans 37.50 + 46.75 + 37.50 m made of prestressed concrete and with variable beam depth. On the basis of numerical simulations, the paper provides author's recommendations for computer modeling of similar bridges.

Keywords: numerical models, prestressed concrete bridges, computer analysis of superstructure, transverse distribution of load

[1]PhD., Eng., Warsaw University of Technology, Faculty of Civil Engineering, Al. Armii Ludowej 16, 00-637 Warsaw, Poland, e-mail: radoslaw.oleszek@pw.edu.pl, ORCID: 0000-0002-1036-329X
[2]Prof., DSc., PhD., Eng., Warsaw University of Technology, Faculty of Civil Engineering, Al. Armii Ludowej 16, 00-637 Warsaw, Poland, e-mail: wojciech.radomski@pw.edu.pl, ORCID: 0000-0002-3404-6109
[3]Msc., Eng., Warsaw University of Technology, Faculty of Civil Engineering, Al. Armii Ludowej 16, 00-637 Warsaw, Poland, e-mail: krzysztof.nowak2@pw.edu.pl, ORCID: 0000-0003-4079-6579

1. Introduction

Static analyses of bridge structures are currently performed using the finite element method (FEM) [1–8]. As we know, it consists in transformation of the analytic problem expressed by differential equations to an algebraic problem [9–15]. This process transforms the mathematical description of the displacement or stress fields, expressed by an infinite number of parameters, into a description expressed by a finite number of parameters located in a limited number of points (nodes) of the structure. In the areas between nodes, the variability of the above fields is determined by functional dependencies, i.e. interpolation functions expressed as the shape functions. In most cases, the user of a computer software has a limited ability to view the exact algorithm of the method programmed in the MES system. The sought static quantities such as displacements, internal forces, stresses, deformations or angles of rotation of the nodes of the discrete system, are determined on the basis of the equilibrium equations of the structural elements in matrix form:

$$\mathbf{K} \cdot \mathbf{u} = \mathbf{P} \tag{1.1}$$

in which: **K** – stiffness matrix characterizing the proportions of stiffness (lengths, moments of inertia, cross-sections) of the elements of the structure, **P** – vector of external loads, **u** – vector of generalized displacements of nodes of the numerical model.

Depending on the geometry of the bridge structures and the required accuracy of calculations, different levels of discretization of these structures are used in their [1, 3, 4, 6, 11, 16–19]. In the design process, beam grillage models (in general denoted e^1, p^2), shell models (in general denoted e^2, p^2) or shell-beam models (denoted $e^1 + e^2$, p^3) are often used. Solid models (in general denoted $e^3 + p^3$) are mostly used in advanced analyses, having frequently a scientific character (e^1 – beam elements, e^2 – plate/shell elements, e^3 – solid elements, p^1 – 1D, the space on a straight line, p^2 – 2D, the space on a plane, p^3 – 3D, the whole space).

In the light of computer analysis methods, the structural model is a broad term that includes the span geometry model, the load model, and the material model. In the paper, the concept of a span geometry model is identified with a numerical model of a structure, otherwise a mechanical model of a structural system, i.e. its static system, and – according to FEM terminology – a discrete model.

It is shown herein that there is an impact of the applied type of the numerical model (i.e., degree of complexity, degree of discretization, accuracy of the model) of the road bridge on the calculated values of bending moments and displacements, which indirectly affects the global safety coefficient of the designed bridge structure. The main purpose of the calculations is to examine the discrepancies of analyzed internal forces and displacements depending on the type of numerical model used. The calculated values are referred to the results taken from the field tests of the existing bridge [20].

2. Description of the bridge structure

The bridge, denoted MS 03, which is the subject of presented below comparative analyze, is a continuous beam structure with three spans 37.50 + 48.75 + 37.50 m made of prestressed concrete [20]. Main dimensions of the structure is shown in Fig. 1. The bridge is located in skew angle of 82.8° and was designed for class “A” according to PN-S-10030:1985 [21].

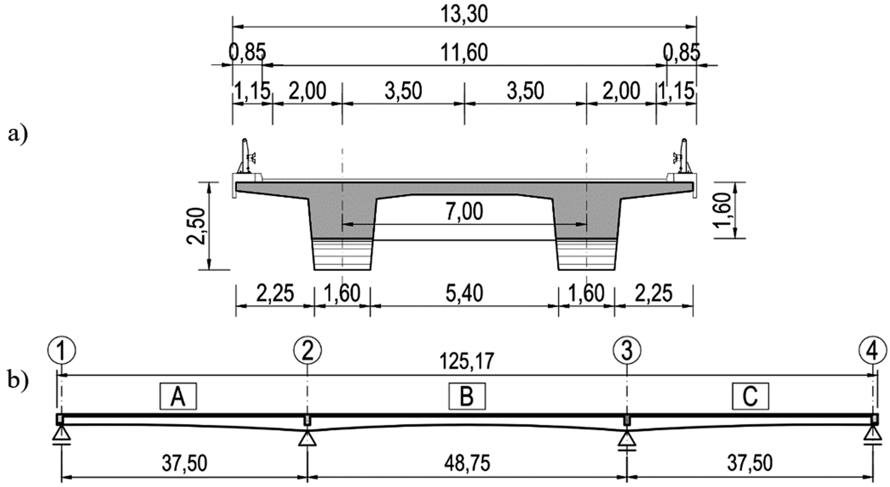

Fig. 1. Geometry of the MS-03 bridge structure (dimensions in [m]): a) cross-section, b) longitudinal section

The main girders are two massive trapezoidal post-tensioned concrete beams with a lateral spacing of 7.0 m. They are made of B50 concrete reinforced with bars of AIIIN steel. The depth of the beams is variable from 1.60 m in the middle of the longest span to 2.50 m over the intermediate supports. The depth of reinforced concrete deck is also variable, from 35 cm in its central part to 47 cm in the fixing place in the massive trapezoidal main beams. The main beams are prestressed with 8 or 9 cables 19Ô150 mm^2 of prestressing steel class R_{vk} = 1860 MPa.

3. Numerical model and bridge loads

The double-beam superstructure of the bridge, without any intermediate crossbeams (i.e., excluding over the support ones), is a typical structural solution of prestressed concrete. Transversal stiffness of the structure can be modelled using FEM in several ways [3, 4, 6–8, 11, 17–19, 22]. It is important that in the systems without intermediate crossbeams, the over support crossbeams have a negligible impact on the distribution of loads on the main beams. The element responsible for the distribution of eccentric loads on the main beams is primarily the deck and to some extent the torsional stiffness of massive trapezoidal beams themselves. In the calculations of this type of spans, the most commonly used are the grillage models (e^1, p^2), occasionally two-dimensional beam-shell structures ($e^1 + e^2$, p^2) or spatial beam-shell structures ($e^1 + e^2$, p^3). In some exceptional cases (e.g. expert studies) entirely shell (e^2, p^2 or e^2, p^3) or solid (e^3, p^3) models are used.

In this work, three models of the bridge in the FEM software SOFiSTiK were applied: beam grillage in two variants (R-1 and R-2, type e^1, p^2) and one beam-shell model (PB, type $e^1 + e^2$, p^2). Comparative calculations of the analyzed bridge were carried out in order that:

– determining the influence of the numerical model type (grillage, beam-shell) on the transverse distribution of the load used during field testing of the bridge and – in consequence – the calculated internal forces and displacements of the structure;
– estimation of the modeling impact of the bridge deck fixing section in main beams on the transverse distribution of the load.

R-1 (Fig. 2a) is a beam grillage model in which all the MES mesh nodes are located in one plane (e^1, p^2). The longitudinal stiffness of the span was modelled with Timoshenko-type beam elements (the influence of shear deformations was taken into consideration) with T-sections and a variable depth $h(x)$. The transverse stiffness of the double main beam system was simulated by weightless transverse beams with cross-sections $b \times h = 100 \times 35$ cm, stretched between the nodes of longitudinal elements (the span of the transverse bands is equal to the spacing of the main beams, i.e. 7.0 m). The impact of the width of the trapezoidal webs of the main beams on the transversal load distribution was not taken into account. The span support was defined by taking the degrees of freedom, according to the bridge bearing scheme, at the "corner" nodes of the grillage model. This does not take into account the actual (eccentric) position of the bearings in relation to the centre of gravity of the main beams.

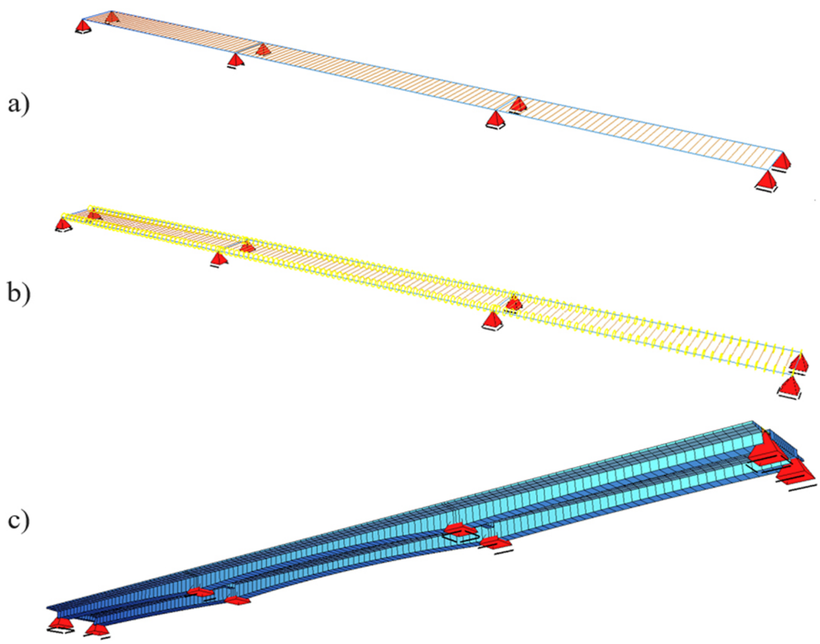

Fig. 2. Visualization of the considered numerical models of the MS-03 bridge superstructure: a) beam grillage model R-1 (e^1, p^2) without sections of fixing the plate in the main beams, b) beam grillage model R-2 (e^1, p^2) with kinematic constraints modeling the plate fixing, c) shell-beam model PB ($e^1 + e^2$, p^2)

R-2 (Fig. 2b) is also a beam grillage model (e^1, p^2), arranged similarly to the R-1 model. In this case, however, the transverse stiffness of the span was modelled in a different way. On the edges of the trapezoidal beams, in the place of the real fixing of the deck in the main beams, additional MES mesh nodes were placed. Between them, transverse beam elements modeling the deck were stretched, taking into account its real span (5.0 m). Additional deck fixing nodes were connected to nodes in the axis of the main beams by means of kinematic constraints concatenating all degrees of freedom (translational and rotational – simulation of a fixed connection of elements). In this way, the influence of the width of the trapezoidal web (the so-called the deck fixing section in the main beams) and the real span of the deck on the behaviour of superstructure in the transverse direction (distribution of loads on the main beams) was taken into account.

PB (Fig. 2c) is a flat beam-shell model ($e^1 + e^2$, p^2) of the structure, in which only trapezoidal web of the main beam and over support crossbeams were modelled with Timoshenko-type beam elements. The cantilevers for the sidewalks and the deck between the main beam were modelled using panels (areas) discretized with Mindlin–Raissner type shell elements (so-called quads). The variable thickness of the cantilevers and the cant (thickening) of the deck slab were taken into account. The deck shell panels were connected to nodes offset from the axis of the main beams, taking into account the real span of the deck. These nodes were connected by kinematic constraints with nodes along the length of the main beams (concatenating all degrees of freedom). Above the main beams projection, additional shell areas with additional geometric feature (without any stiffness) were defined to unify the method of load distribution.

All the models take into account the variable height of the section along the length of the spans according to the function of the circle equation. The support points simulating the bearings are represented by the corner nodes of the main beams (the eccentricity of the bearings relative to the upper surface of the span was not taken into account).

The evaluation of the distribution of loads to the main beams was carried out taking into account the load of the IVECO Trakker Man vehicles (four-axle tipper trucks with the total mass of each equal to 38 tonnes) used during the field tests of the bridge [20]. The front axle loads are 80 kN and the rear axles 110 kN. In the performed numerical simulations were considered (Fig. 3):

– Scheme I – symmetrical setting sequences of 3 vehicles in the cross section of the bridge superstructure (total width of the road) and two vehicles in the longitudinal direction (a total of $2 \times 3 = 6$ vehicles), in terms of assessing the convergence of calculations with the results of load tests [20],
– Scheme II – sequences of asymmetric settings, one vehicle in the transverse direction (two in the longitudinal direction) at the edge of the roadway, which made it possible to investigate the transverse load distribution depending on the superstructure modelling method.

The loads on the axles of trucks have been represented the surface area loads, reduced to the area of contact between the tire and the road pavement with dimensios of 20×60 cm. The total load on individual wheels of the front axles was $P_p = 40$ kN, and the rear axle $P_t = 55$ kN. The envelope of vehicle settings along the length of the structure was simulated using a package of static loads placed on the spans with a assumed step (so-called loadcases). In the case of the R-1 and R-2 (e^1, p^2) grillage models, wheel load surface loads were modelled using the "free loads projected onto the model" option of the SOFiLOAD module. The conversion of these

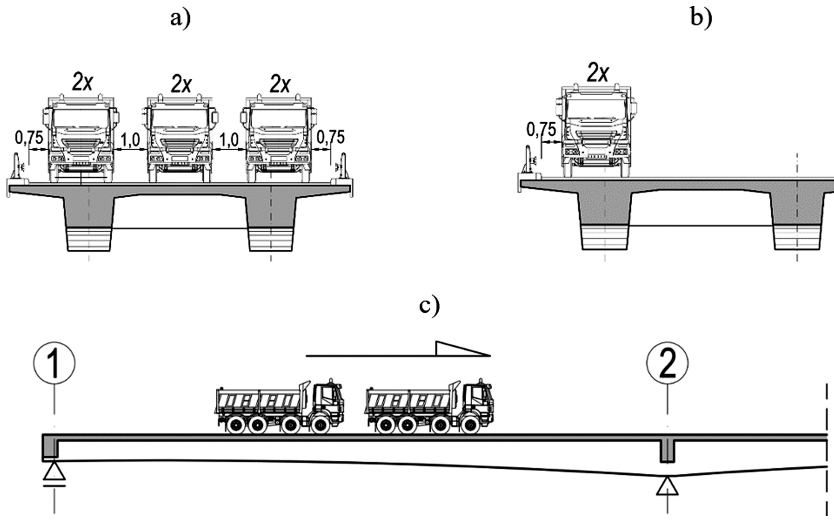

Fig. 3. Considered loads of the structure: a) Scheme I – symmetrical, b) Scheme II – asymmetrical (one-sided overload of the span)

loads into equivalent forces applied to the nodes of the model was carried out using an organic algorithm of the SOFiSTiK system.

In the case of the shell-beam model ($e^1 + e^2$, p^3), the total load on the wheels of the front and rear axles was distributed over the contact area of the tire with the road pavement and applied to the surface elements (quads) of the panels modelling the deck and the cantilevers for the sidewalks.

4. Discussion of the results of calculation

Due to the type of the superstructure of the analyzed bridge (two-girder, three-span continuous beam), the envelope of vertical displacements uz along the length of the structure and the bending moments My in the main beams from individual live load cases were considered as static values enabling the evaluation of the applied numerical models.

The results of elastic deflections calculated on the basis of the numerical models (R-1, R-2, PB) were compared with the values of deflections obtained in the field tests-of the bridge (Table 1). The superstructure was designed as prestressed. Therefore, in calculations and load tests, the superstructure works as non-cracked in phase I, which means that the cross-sections of the main beams are characterized by full (unreduced) bending stiffness $E_b J_y$.

In general, for prestressed structures, the design is usually carried out in the elastic range. Mathematical packages, such as the Matlab program, can be successfully applied for linear statics calculations. An example of using this program in engineering calculations is presented in [23]. In paper [24], the authors presented an example of a non-linear structural analysis for a reinforced concrete and prestressed concrete load-bearing structure of a bridge.

Table 1. Calculated (u_{zobl}) and measured (u_{zpom}) deflections of the bridge superstructure – Scheme I

Span		I		II		III	
Main beam		A / left	B / right	A / left	B / right	A / left	B / right
u_{zpom} [mm]		11.18	10.75	17.63	17.23	11.11	11.08
u_{zobl} [mm]	Model R-1	13.13	13.14	19.58	19.60	12.95	12.99
	Model R-2	13.08	13.09	19.60	19.63	12.91	12.99
	Model PB	13.68	13.68	20.50	20.50	13.66	13.66
$\frac{u_{zpom}}{u_{zobl}}$	Model R-1	0.851	0.818	0.900	0.879	0.858	0.853
	Model R-2	0.855	0.821	0.899	0.878	0.861	0.853
	Model PB	0.817	0.786	0.860	0.840	0.813	0.811

Based on the data in Table 1, it should be stated that in the superstructure, the theoretical deflections calculated on the basis of the R-1 and R-2 grillage models (e^1, p^2) are the closest to the deflections of the real superstructure taken from its field tests. The relevant differences varies in the range of 10÷18%. On the other hand, the u_z displacements obtained based on the PB model ($e^1 + e^2$, p^3), which is the most faithful representation of the superstructure in the mathematical and physical sense, correspond to a slightly lesser extent with the measured values, with differences of 14÷21%. However, this does not mean that the beam-shell models are flawed. This reveals about 10÷20% increase in the stiffness of the superstructure, in relation to the value obtained on the basis of the static and strength parameters of the span cross-section (modules of elasticity of concrete, moments of inertia, etc.). This phenomenon is typical for concrete bridges submitted to load tests [16, 25]. It may be caused, among others, by: obtaining an increased modulus of elasticity of the concrete of the real structure, in relation to the standard value adopted in the design, not taking into account the moments of inertia of concrete sections in phase I (not cracked) including the section of reinforcement and prestressing reinforcement at the stage of static calculations (according to the authors, the impact of this is about 5%) and the influence of the stiffness of sidewalk slabs, guard rails and even the road pavement. Due to the highest accuracy, the PB model should be considered as a reference (basic) model in relation to the other two grillage models (R-1 and R-2).

The discrepancies of the deflections calculated using FEM models and determined on the basis of measurements during the field tests reach about 20%, which is a typical value for similar prestressed structures built in Poland (cf. e.g. [16, 25]). Deflections of the bridge superstructure obtained from all the models used (i.e., R-1, R-2, PB) are similar with differences (u_{zmax}/u_{zmin}) in the range of 4.6÷5.8%, on average 5%. It follows that the considered computer models, despite of the different degree of discretization (accuracy) adequately represent the stiffness of the structure. On the other hand, the differences of up to 20% between the calculations and the results of field tests result from the simplifications that are routinely made in modeling this type of structure. These simplifications consist in omitting the influence of the sidewalk slabs, guard rails and the road pavement on the stiffness of structure as well as adopting in the calculations the standard value of the modulus of elasticity of concrete E_b and not taking into account the influence of the stiffness of the reinforcement on the geometrical characteristics of the cross-sections.

Fig. 4 presents an envelope diagram of vertical displacements u_z along the length of the structure, caused by a live load (a set of tipper trucks). In the case of symmetrical load of the bridge superstructure with six vehicles (Scheme I), the largest deflection values occur in the PB model (shell-beam, class $e^1 + e^2$, p^3). In the R-1 and R-2 grillage models, the deflection lines have virtually identical course, with deflection values 5% lower than obtained from the PB model. On the other hand, the simplest grillage model R-1 is characterized by the largest deflections from one-sided loaded the structure (Scheme II – setting of two tipper trucks). The elastic maximum deflections of the bridge three spans are 6.0 mm, 8.4 mm, 5.8 mm, respectively. With such a load scheme, the R-2 grillage model (modelled sections of deck fixing in the girders and the beam-shell model behave similarly, with maximum deflections of bridge spans of 5.4 mm, 7.8 mm, 5.3 mm, respectively.

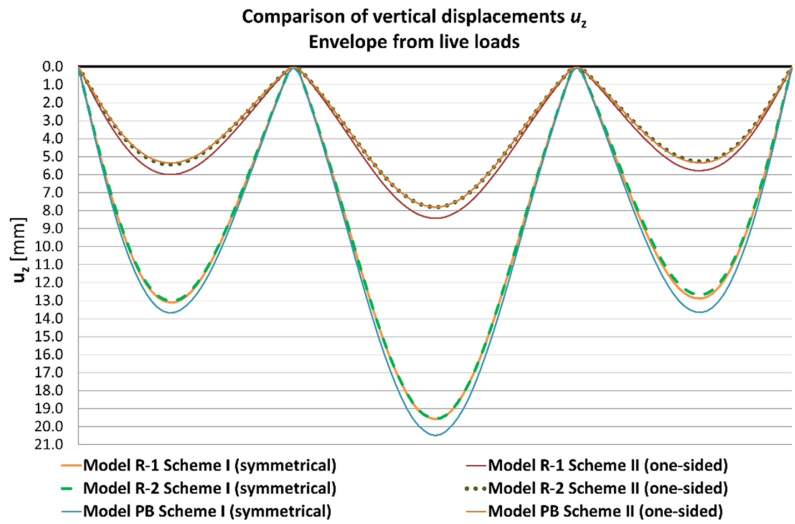

Fig. 4. Comparison of the envelope of vertical displacements u_z due to vehicle loads

The vertical displacements u_z of the the bridge superstructure calculated using the R-2 and PB models are smaller than in the R-1 model by about 7÷11%. This is due to the fact that in the simplest grillage model R-1, the section of fixing the deck in the main beams is not taken into account, and its span in this model is equal to the distance between the centers of gravity of the main beams. The result of not taking into account the width of the trapezoidal cross-section of the main beam is a slight overstatement the span of the deck in the model and thus underestimation of the relative stiffness of the element $E_b J_y / L$. This results in the greatest overloading effect of the directly loaded girder. In the analyzed bridge structure, due to the lack of intermediate crossbeams, the element implementing the transverse distribution of the load is the deck slab only (cf. [17–19]).

From the point of view of modelling structures similar to the analysed bridge, the key issue is the transverse distribution of the life load on the main beams, which generates internal forces (or stresses) in them, which are the basis for dimensioning. Numerical models, depending on the class (method of discretization, type of finite elements used, etc.) and the degree of detail, can distribute loads to girders with different accuracy.

Indirectly, for the design of prestressed concrete main beams, the envelope of the bending moments M_y in these elements is used, caused by the live load moving along the bridge. In the case of the presented herein FEM models, on the basis of a comparison of diagrams of the bending moments produced by one-sided loading of the bridge deck (*Scheme II*), it can be concluded how a relevant model implements the transverse load distribution between the girders (Fig. 5).

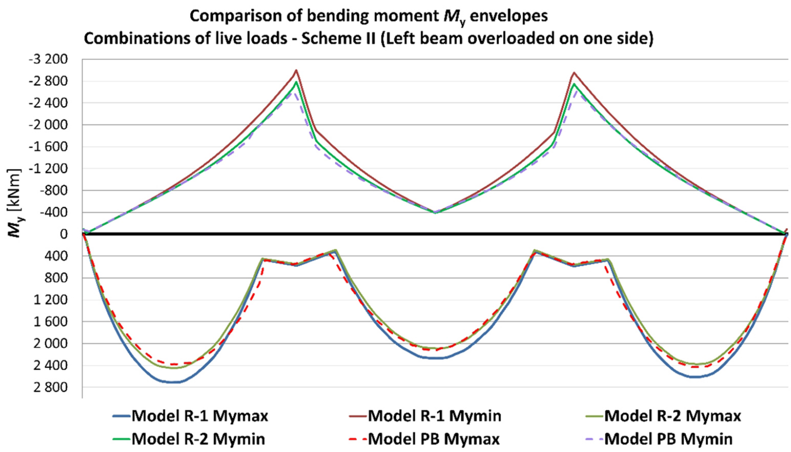

Fig. 5. Comparison of the envelope of bending moments M_y in the main beams obtained using three numerical models of the bridge

In the case of the R-1 and R-2 (e^1, p^2) grillage models, the values of the bending moments, related to the T-sections of the main beams, were taken directly from the models. In the PB shell-beam model ($e^1 + e^2$, p^2), the crossbeams over support and the trapezoidal main beams were modelled only, using beam elements. The cantilevers and the deck were modelled using panels (areas) discretized with Mindlin–Reissner shell elements. For this reason, it was necessary to transform the unit internal forces (in principle normal stress fields) into an integrated form. The SIR module function was used to integrate σ_x normal stress from the beam and shell elements of the model.

With respect to the dimensioning of the prestressed main beams, the values of bending moments M_y from live loads in critical cross-sections (i.e., in span and support zones) are important. A comparison of the extreme values of the moments M_y, which were obtained with one-sided deck loading (Scheme II) using three numerical models (R-1, R-2, PB) is presented in Table 2.

On the basis of the comparison of the values of bending moments M_y in the integrated form taken from the numerical models (R-1, R-2, PB), it can be concluded that the influence of the model class type is moderately significant. The differences in extreme (over support or span) bending moments M_y at one-sided loading of the bridge deck (Scheme II) vary in the following ranges (cf. Table 2):

– 7.6÷19.1% in the case of the simplest grillage model R-1, which overestimates the values of bending moments in relation to the PB model,
– in the case of R-2 grillage model which overestimates the values of span moments in the range of 4.7÷9.1%, while underestimates the support values of bending moments by 0.9÷10.7%.

Table 2. Comparison of extreme bending moments M_y – Scheme II

Numerical model						
		Span A	Support 2	Span B	Support 3	Span C
Symbol	Type	M_{max} [kNm]	M_{min} [kNm]	M_{max} [kNm]	M_{min} [kNm]	M_{max} [kNm]
R-1	e^1, p^2	2712	–2999	2271	–2955	2612
		+14.1%	+19.1%	+7.6%	+17.4%	+7.6%
R-2		2446	–2787	2090	–2746	2377
		+4.7%	+10.7%	–0.9%	+9.1%	–2.1%
P-B	e^2 (+e^1), p^2	2337	–2518	2110	–2516	2427
Note: Below the bending moment value, the percentage difference from the reference value (Model PB) is given, the sign " ? " means that the absolute value of the moment has decreased compared to the comparative model.						

In eight cases, the differences in calculation results are less than the acceptable deviation limit in the estimation of static quantities in engineering structures equal to 15%, which can be considered as insignificant. In two cases of the simplest model R-1, the discrepancies are greater (17.4% and 19.1%), so that the impact of the model type on the calculation results can be considered moderately significant (differences in the range of 15÷25%). The method of modelling the section of the deck fixing in the main beams is exposed here, i.e., the influence of the width of the beam on the effective span and stiffness of the deck $E_b J_y / L$, has a relatively important effect on the transverse load distribution on the main beams. However, the analysis shows that modeling of the joints between the main beams and transversal beams simulating the deck, has rather moderately significant impact on the calculation results.

5. Final remarks and conclusions

Based on the comparative calculations of the MS-03 bridge, using three numerical models (beam grillage R-1 and R-2 and shell-beam PB), it should be stated that the influence of the FEM model type is moderately significant, i.e. the discrepancies of the estimated values of the bending moments are in the range of 15÷25%. In the case of bending moments M_y from live loads with one-sided loading of the bridge deck, due to the transverse load distribution, the discrepancies between the models reach a maximum of 19.1%.

The application of models with different degrees of discretization causes that the differences in results determined from the case when the live loads set is asymmetrically located in the cross-section of the bridge superstructure, exceed 15%, i.e. the value usually considered in engineering calculations as the acceptable deviation limit. However, when these loads are combined with other actions (e.g. dead-load of structure and bridge detail elements), the effective influence on the design output will be less than 15%. Constructing structures with symmetrical cross-sections of the spans to some extent eliminates the imperfections of their modelling using FEM models of various types herein presented.

In the case of the R-1 and R-2 beam grillage models, the effect of taking into account or not the width of the trapezoidal main beam (the so-called the deck fixing section) on the effective deck span and the transverse load distribution of the one-sided (asymmetrical) load, is associated with bending moment discrepancies in the range of 7.6÷10.9%. The differences in the value of M_y from live loads do not exceed 15%, what means that the influence of the method of modeling the width of the beams (or omitting the modelling of their width) is insignificant in the analyzed herein bridge structure. The impact of this aspect of modeling spans with the wide main beams on the final results (e.g., M_y envelopes) will be even smaller after taking into account the remaining loads acting normally on the bridge.

In the case of performing static calculations at the stage of designing of the structures similar to the analyzed bridge, the authors recommend using the grillage models that take into account the width of the main beams and the real span of the deck slab for transverse load distribution. So-called the fixing section of the deck in the main beam (width of the trapezoidal beam) can be easily represented using beam finite elements with increased stiffness or kinematic constraints, depending on the software available for the designer. In this paper, the model of this type overestimates the values of bending moments to about 10% in relation to the beam-shell reference model, which is more advanced in terms of construction theory and gives more safe static values due to their overestimation compared with the other models and some results of the field test of the bridge. The labor consumption of its implementation is relatively small, and the size of the numerical task and the calculation time (e.g., determining the envelope of static quantities) are small. This is important in the process of designing prestressed superstructures due to the need for multi-variant calculations for different routes of prestressing cables. Analysis results in the form of integrated values of internal forces are referenced to beams, which facilitates their interpretation and design of the structure based on design standards. In beam elements, it is easier and more reliable to model the effect of prestressing cables on the structure (equivalent load method), compared to shell elements.

References

[1] Ł. Chełstowski, R. Oleszek, and W. Radomski, "The modelling possibilities of two beams spans concrete bridges", *Inżynieria i Budownictwo*, no. 7–8, pp. 422–428, 2013 (in Polish).

[2] J. Chróścielewski, M. Miśkiewicz, and Ł. Pyrzowski, *Introduction to FEM modelling in the SOFISTIK software*. Gdańsk: Wydawnictwo Politechniki Gdańskiej, 2016 (in Polish).

[3] J. Chróścielewski, T. Galewski, A. Kozakiewicz, M. Malinowski, and K. Żółtowski, "Consequences of the method of modelling a bridge on the example of a prestressed road viaduct", presented at 21th International Conference on Structural Failures, Szczecin–Międzyzdroje, 2003 (in Polish).

[4] E.C. Hambly, *Bridge deck behavior*. London: Chapman and Hall, 1976.

[5] A. Jukic and K. Ekfeldt, *Concrete Bridge Design with FEM. A comparative analysis between 3D shell and 2D frame models*. Göteborg: Chalmers University of Technology, Department of Civil and Environmental Engineering, 2012.

[6] J. Kmita, J. Bień, and Cz. Machelski, *Computer-aided design of bridges*. Warszawa: WKŁ, 1989 (in Polish).
[7] E.J. O'brien and L. D. Keogh, *Bridge deck analysis*. London: E and FN Spon, 1999.
[8] G. A. Rombach, *Finite element design of concrete structures*. London: Thomas Telford Publishing, 2004.
[9] K. Bathe, *Finite element procedures in engineering analysis*. Englewood Cliffs: Prentice-Hall, Inc., 1996.
[10] R.D. Cook, D.S. Malkus, M.E. Plesha, and R.J. Witt, *Concepts and applications of finite element analysis*. New York: John Wiley and Sons, 1989, 2002.
[11] J. Grycz, "Static analysis methods of beam-plate models", *Budownictwo betonowe, vol. XIV*. *Mosty cz. 2*. Arkady, 1973 (in Polish).
[12] T.J.R. Hughes, *The finite element method. Linear static and dynamic finite element analysis*. Mineola, New York: Dover Publications, 2000.
[13] G. Rakowski and Z. Kacprzyk, *The finite element method in structural mechanics*. Warszawa: Oficyna Wydawnicza Politechniki Warszawskiej, 2005 (in Polish).
[14] J. Szmelter, M. Dacko, S. Dobrociński, and M. Wieczorek, *The finite element method in structural statics*. Warszawa: Arkady, 1979 (in Polish).
[15] O.C. Zienkiewicz, R.L. Taylor, and J.Z. Zhu, *The finite element method: its basis and fundamentals*. Amsterdam: Elsevier, Butterworth-Heinemann, 2005.
[16] P. Łaziński, "Modeling procedure of real structures in the form of a certain type of bridge structures", PhD thesis, Silesian University of Technology, Gliwice, 2009 (in Polish).
[17] Cz. Machelski, "Parameters of load transverse distribution across bridges", *Roads and Bridges – Drog i i Mosty*, vol. 13, no. 2, pp. 131–143, 2014, doi: 10.7409/rabdim.014.009.
[18] Cz. Machelski, "Transverse distribution of loading in bridges with two capital girders", *Inżynieria i Budownictwo*, no. 9, pp. 490–493, 2003 (in Polish).
[19] Cz. Machelski, J. Onysyk, and J. Biliszczuk, "Analysis of working the double-girder superstructures made of prefabricated elements type U", *Inżynieria i Budownictwo*, no. 6, 1997 (in Polish).
[20] *Results of the field tests of the MS-03 bridge over the Tyśmienica River at km 4+114.48 of the DK19 road*. Bielsko-Biała: Design and Research Office "System", 2011 (in Polish).
[21] PN-S-10030:1985 Bridge objects. Loads. PKN, 1985 (in Polish).
[22] S. Timoshenko and S. Woinowsky-Krieger, *Plate and shell theory.* Warszawa: Arkady, 1962 (in Polish).
[23] P. Hawryszków and B. Czaplewski, "Application of Matlab software in static calculations of bridge structures", *Archives of Civil Engineering*, vol. 68, no. 1, pp. 299–317, 2022, doi: 10.24425/ace.2022.140169.
[24] T.V.T. Thi and Q.V.T. Bich, "Nonlinear analysis for proposing limit state criteria of reinforced concrete road bridge superstructures", *Archives of Civil Engineering*, vol. 69, no. 1, pp. 495–512, 2023, doi: 10.24425/ace.2023.144185.
[25] P. Łaziski, M. Salamak, and T. Owerko, "Field tests of bridges in Poland based on selected examples", presented at 7th Polish Conference "Konstrukcja i Wyposażenie Mostów", Wisła, 2015 (in Polish).

Wpływ klasy modelu numerycznego mostu z betonu sprężonego na wyznaczane wielkości statyczne

Słowa kluczowe: modele numeryczne mostu, mosty z betonu sprężonego, analiza komputerowa przęsła, rozdział poprzeczny obciążenia

Streszczenie:

Analizy statyczne konstrukcji mostowych są obecnie wykonywane metodą elementów skończonych (MES) [1–8]. W zależności od geometrii konstrukcji i wymaganej technicznie dokładności obliczeń, w projektowaniu i analizie statycznej obiektów mostowych stosuje się różne poziomy dyskretyzacji tych struktur [9–15]. W projektowaniu często stosowane bywają modele rusztowe (klasy e^1, p^2), powłokowo-belkowe (klasy $e^1 + e^2$, p^3) ewentualnie powłokowe (klasy e^2, p^2). Modele bryłowe stosuje

się przeważnie w zaawansowanych analizach, także o charakterze naukowym. W pracy wykazano, że istnieje wpływ przyjętej klasy modelu numerycznego wiaduktu drogowego (stopień skomplikowania, dyskretyzacji, dokładność modelu) na otrzymywane wartości momentów zginających i przemieszczeń oraz ich różnice, co pośrednio wpływa na współczynnik globalnej rezerwy bezpieczeństwa projektowanej konstrukcji mostowej. Głównym celem obliczeń było zbadanie rozbieżności oszacowanych wielkości statycznych między modelami o różnej dokładności, a przeprowadzone symulacje komputerowe mają charakter eksperymentu numerycznego. Wykonane obliczenia nawiązywały do wyników uzyskanych podczas badan odbiorczych obiektu pod próbnym obciążeniem [20]. Przedmiotem analiz porównawczych byłmost MS-03, który jest konstrukcją płytowo-belkową z betonu sprężonego o schemacie belki ciągłej trójprzęsłowej o rozpiętościach przęseł37,50 + 48,75 + 37,50 m i zmiennej wysokości belek. W pracy wykonano trzy modele obiektu w środowisku MES SOFiSTiK: rusztowy w dwóch wariantach (R-1 i R-2 klasy e^1, p^2) i mieszany powłokowo-belkowy (PB, klasy e^1+e^2, p^2). Obliczenia porównawcze badanego obiektu przeprowadzono w celu określenia wpływu klasy modelu numerycznego (rusztowy, mieszany belkowo-powłokowy) na rozdziałpoprzeczny obciążenia użytego podczas badań odbiorczych (część obciążenia przypadająca na pojedynczy dźwigar) i wygenerowane wielkości statyczne (momenty zginające, przemieszczenia) oraz oszacowania wpływu sposobu modelowania odcinka utwierdzenia płyty pomostowej w środnikach dźwigarów, pełniącej rolę elementu stężającego belki nośne na rozdziałpoprzeczny obciążenia.Na podstawie przeprowadzonych obliczeń porównawczych mostu, za pomocą trzech modeli numerycznych (rusztowych R-1 i R-2 oraz powłokowo-belkowego PB) należy stwierdzić, że wpływ klasy modelu MES (stopnia dyskretyzacji) jest średnio znaczący, tj. rozbieżności oszacowanych wielkości statycznych zawierają się w przedziale 15÷25%. W przypadku momentów zginających M_y od obciążeń ruchomych przy jednostronnym przeciążeniu przęsła, z uwagi na rozdziałpoprzeczny obciążenia, rozbieżności wyników między odwzorowaniami osiągają maksymalnie 19,1%. Zastosowanie modeli o różnym stopniu dyskretyzacji powoduje, że różnice wyników od obciążeń ruchomych ustawionych niesymetrycznie w przekroju przęsła przekraczają 15%, a więc wartość zwyczajowo uznawaną w obliczeniach inżynierskich za akceptowaną granicę błędu. W przypadku odwzorowań rusztowych R-1 i R-2 efekt uwzględnienia lub nieuwzględnienia szerokości środników belek trapezowych (tzw. odcinek utwierdzenia płyty pomostu) na efektywną rozpiętość płyty pomostu oraz rozdziałpoprzeczny obciążenia jednostronnego (niesymetrycznego) wiąże się z rozbieżnościami momentów zginających w zakresie 7,6÷10,9%. Różnice wartości M_y od obciążeń ruchomych nie przekraczają 15%, co oznacza, że wpływ sposobu modelowania szerokości środników (lub pominięcie odwzorowania ich szerokości) jest w omawianej konstrukcji mało znaczący. Wpływ tego aspektu modelowania przęseło szerokich środnikach na ostateczne wyniki (np. obwiednie M_y) będzie jeszcze mniejszy po uwzględnieniu pozostałych obciążeń obiektu. Na podstawie przeprowadzonych symulacji numerycznych w pracy podano rekomendacje autorskie dotyczące komputerowego modelowania podobnych obiektów.

Received: 2023-02-06, Revised: 2023-07-25