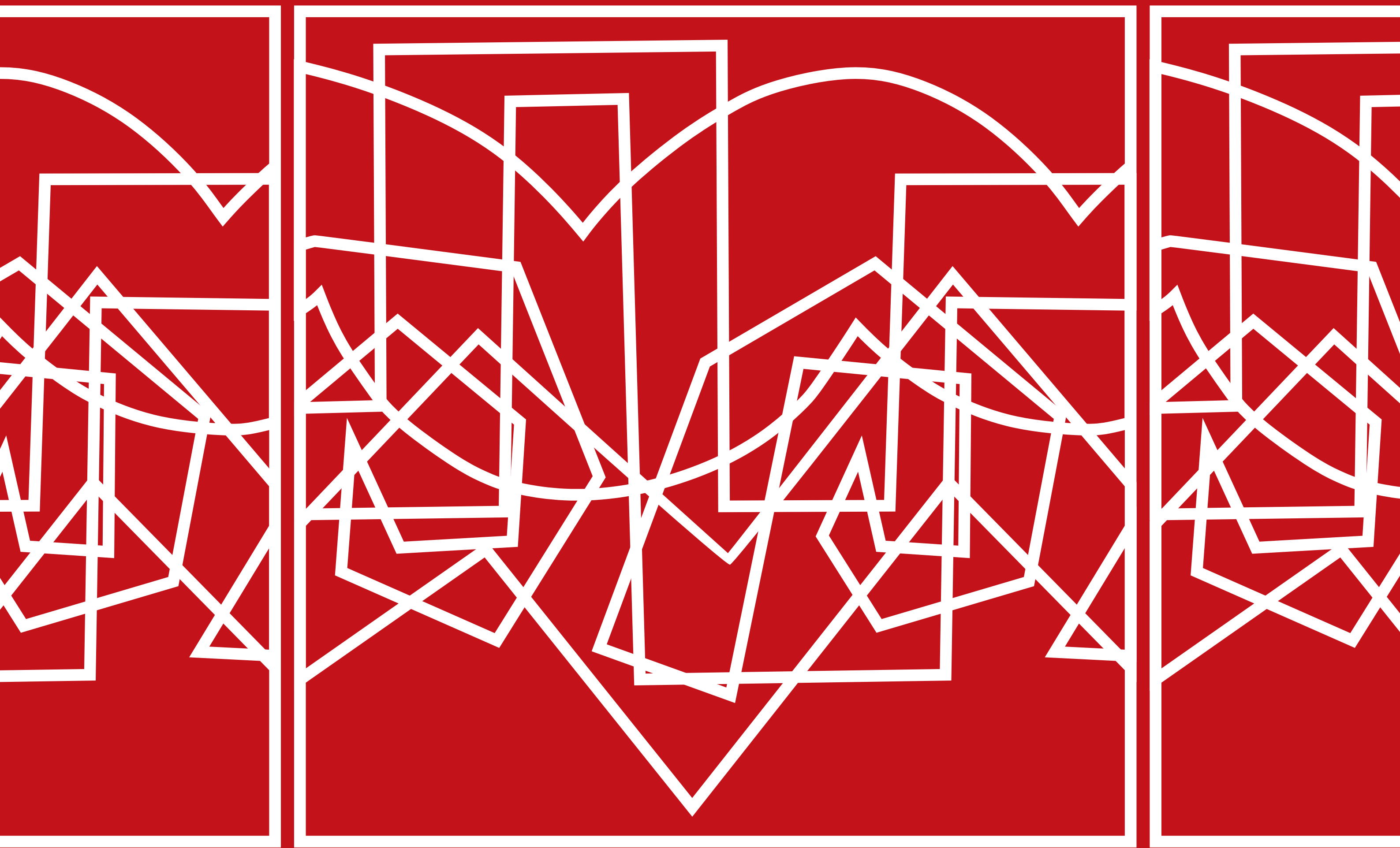


ПРИРОДНИ КАТАСТРОФИ И КОНСТРУКЦИИ
NATURAL HAZARDS AND STRUCTURES

ДГКМ

ДРУШТВО НА ГРАДЕЖНИ КОНСТРУКТОРИ НА МАКЕДОНИЈА

MASE

MACEDONIAN ASSOCIATION OF STRUCTURAL ENGINEERS

17 МЕЃУНАРОДЕН СИМПОЗИУМ
INTERNATIONAL SYMPOSIUM

ОХРИД, МАКЕДОНИЈА
OHRID, MACEDONIA
4 - 7 октомври 2017
october, 4th - 7th , 2017

MASE ДГКМ

Macedonian Association of Structural Engineers
Друштво на градежните конструктори на Македонија

Proceedings
Зборник на трудови

17[th] International Symposium
17[ти] Меѓународен симпозиум

Ohrid, Macedonia, 4 – 7 October 2017
Охрид, Македонија, 4 – 7 Октомври 2017

PROCEEDINGS
OF THE 17th INTERNATIONAL SYMPOSIUM OF MASE
ЗБОРНИК НА ТРУДОВИ
17ТИ МЕЃУНАРОДЕН СИМПОЗИУМ НА ДГКМ

Publisher:
MASE - Macedonian Association of Structural Engineers
Faculty of Civil Engineering, Blvd. Partizanski odredi No. 24 P.Box. 560,
1000 Skopje, Republic of Macedonia
e-mail: mase@gf.ukim.edu.mk; web-site: www.mase. gf.ukim.edu.mk

Издавач:
ДГКМ - Друштво на Градежни Конструктори на Македонија
Градежен Факултет, бул. Партизански одреди бр. 24 П.Ф. 560,
1000 Скопје, Република Македонија
e-mail: mase@gf.ukim.edu.mk; web-site: www.mase.gf.ukim.edu.mk

Editor: **Meri Cvetkovska, President of MASE**

За издавачот: **Мери Цветковска, Претседател на ДГКМ**

Executive Committee of MASE and
Organizing Committee of the 17th International Symposium of MASE:
Meri Cvetkovska, Petar Cvetanovski, Čedomir Teodosievski, Ana Trombeva-Gavriloska, Roberta Apostolovska, Darko Nakov, Andrea Serafimovski, Sergej Churilov, Imeri Selman, Ilija Markov, Vladimir Vitanov, Koce Todorov, Denis Popovski, Marijana Lazarevska, Marija Vitanova

Претседателство на ДГКМ и
Организационен одбор на 17тиот Меѓународен симпозиум на ДГКМ:
Мери Цветковска, Петар Цветановски, Чедомир Теодосиевски, Ана Тромбева-Гаврилоска, Роберта Апостоловска, Дарко Наков, Андреа Серафимовски, Сергеј Чурилов, Имер Селман, Илија Марков, Владимир Витанов, Коце Тодоров, Денис Поповски, Маријана Лазаревска, Марија Витанова

Technical staff for the Symposium:
Mile Partikov, Cvetanka Chifliganec, Simona Bogoevska, Kristina Milkova, Milica Jovanoska, Riste Volčev, Nikola Postolov, Vladimir Damjanovski, Marija Docevska, Elena Cvetkovska, Maja Mateska, Aleksandra Cubrinovska

Техничка служба на Симпозиумот:
Миле Партиков, Цветанка Чифлиганец, Симона Богоевска, Кристина Милкова, Милица Јованоска, Ристе Волчев, Никола Постолов, Владимир Дамјановски, Марија Доцевска, Елена Цветковска, Маја Матеска, Александра Чубриновска

Grafical design of cover page and Symposium poster:
Mitko Hadzi Pulja, Betim Zeqiri
Faculty of Architecture, UKIM, Skopje

Графички дизајн на корицата и плакатот на Симпозиумот:
Митко Хаџи Пуља, Бетим Зекири
Архитектонски факултет, УКИМ, Скопје

e-book:
електронско издание: ISBN 978-608-4510-32-1

ДГКМ
ДРУШТВО НА ГРАДЕЖНИТЕ КОНСТРУКТОРИ НА МАКЕДОНИЈА

Партизански одреди 24,
П.Фах 560, 1001 Скопје
Македонија

MASE
MACEDONIAN ASSOCIATION OF STRUCTURAL ENGINEERS

Partizanski odredi 24,
P. Box 560, 1001 Skopje
Macedonia

SE -13

mase@gf.ukim.edu.mk
http://mase.gf.ukim.edu.mk

Душан МИЛОЈЕВИЌ[1], Мирослав МАРЈАНОВИЌ[2], Мира ПЕТРОНИЈЕВИЌ[3]

ДИНАМИЧКА АНАЛИЗА НА АБ МОСТ: ГРЕДНИ И ПЛОЧЕСТИ МОДЕЛИ НА ПЛАТФОРМАТА

РЕЗИМЕ

Во овој труд е извршена споредба на добиените резултатите со користење на различни нумерички модели за динамичка анализа на повеќеброден мост во кривина преку реката Нишава. Мостот е лоциран во VII сеизмичка зона. Претпоставеното забрзување на тлото е 0,1g. Иако пристапот за проектирање според капацитетот е пошироко прифатен во современото конструктивно проектирање на мостово, неговата примена во потребна со средна и ниска сеизмичка активност е непотребна. Поради тоа, нумеричката анализа е спроведена користејќи мултимодална спектрална анализа во SAP2000. Најпрво, платформата на мостот е моделирана користејќи гредни елементи, додека во вториот случај платформата е моделирана користејќи плочести елементи. Добиените резултати се презентирани и главните разлики помеѓу двата модели се дискутирани.

Клучни зборови: АБ мост, сеизмичка анализа, мултимодална спектрална анализа

Dušan MILOJEVIĆ[1], Miroslav MARJANOVIĆ[2], Mira PETRONIJEVIĆ[3]

DYNAMIC ANALYSIS OF RC BRIDGE: BEAM VERSUS SHELL DECK MODEL

SUMMARY

In this paper a comparison between the results obtained by using different numerical models for dynamic analysis of multi-span curved bridge over river Nišava is presented. Bridge is located in the VII seismic zone. Assumed acceleration is 0.1g. Although the capacity design approach has become a widely accepted in contemporary structural design of bridges, its application in the case of moderate and low-level earthquake activity is unnecessary. Due to that, the numerical analysis is carried out using the multimodal response spectrum analysis in SAP2000. Firstly, the bridge deck is modeled by using beam elements, while in the second case the deck is modeled by using shell elements. The obtained results are presented and main differences between these two models are discussed.

Keywords: RC bridge, seismic analysis, response spectrum multi modal analysis

[1] BSc, Faculty of Civil Engineering, University of Belgrade, Republic of Serbia, dusan230293@gmail.com
[2] Assoc. Prof. PhD, Faculty of Civil Engineering, University of Belgrade, Republic of Serbia, mmarjanovic@grf.bg.ac.rs
[3] Prof. PhD, Faculty of Civil Engineering, University of Belgrade, Republic of Serbia, pmira@grf.bg.ac.rs

1. INTRODUCTION

In recent years, increasing attention has been paid to the analysis of the earthquake impact on bridges. The reason for this is the collapse of bridges that have occurred in areas with high seismicity (Priestley, Seible and Calvi, 1996). EU has adopted standard EN 1998-2 (2006) related to the earthquake resistant design of bridges. The basic design philosophy of this Standard is to achieve the non-collapse requirement of the structure with appropriate reliability. According to that, the behaviour of bridge under the design seismic action shall be either ductile or limited ductile/essentially elastic, depending on the seismicity of the site. Standard prescribes methods of analysis, verification of bearing capacity and construction detailing for this purpose. The proposed method of analysis is an equivalent linear method using the behaviour factor q. It is defined globally for the entire structure and reflects its ductility capacity i.e. the capability of the ductile members to withstand, with acceptable damage but without failure, seismic actions in the post-elastic range (Priestley, Seible and Calvi, 1996).

In Serbia, even now, most engineers consider the dynamic bridge analysis unnecessary and EN 1998-2 proposal as an excessive requirement. This is supported by the fact that until now there has been no case of the bridge damage due to the seismic activity in Serbia. Therefore, only the linear methods of dynamic analysis of bridges have been used, so far. The main dilemma is: What type of the finite elements should be adopted in the numerical modelling of bridge superstructure?

Proposed study attempts to solve this dilemma comparing the dynamic responses of the reference bridge obtained by using two different finite element models in SAP2000 (SAP2000, 2006). In the first one (beam model), the bridge deck is modelled by using beam element, while in the second one (shell model) a certain numbers of shell elements are used for deck modelling. The analysis is carried out using multimodal spectral analysis. The member forces and displacements are estimated by combining the responses of individual modes using SRSS method. The obtained results and detailed discussion are presented in the following section.

2. REFERENCE BRIDGE STRUCTURE

The Nišava bridge was selected as the reference case for this analysis. It is a seven-span structure of 232.2 m total length, presented in Figure 1. The deck is curved-in-plane, with radius of curvature R=540m and longitudinal slope of 1.82%. It is supported by six RC piers S1-S6, of rectangular hollow section, 6.0×1.3m and unequal clear heights of S1=17.14m, S2=17.51m, S3=16.82m, S4=16.11m, S5=16.07m and S6=9.95m.

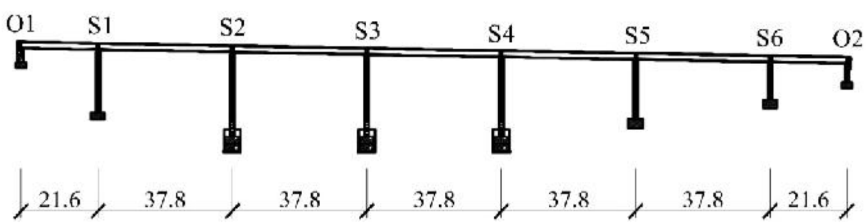


Fig. 1. Structure layout in longitudinal direction

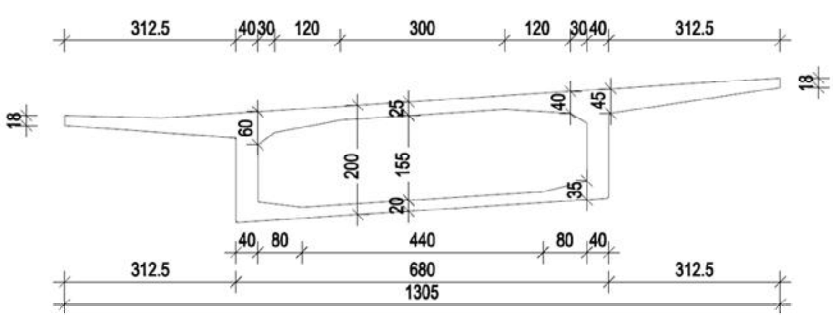


Fig. 2. Cross section of the bridge deck

The RC concrete deck has a box-section, 13.05m wide and 200cm high, with transverse inclination of 6.5%. (Figure 2). In the longitudinal direction, the deck webs and bottom plate have variable thickness. The deck webs thickness vary from 40 to 70cm, while the bottom plate thickness vary from 20-40cm.

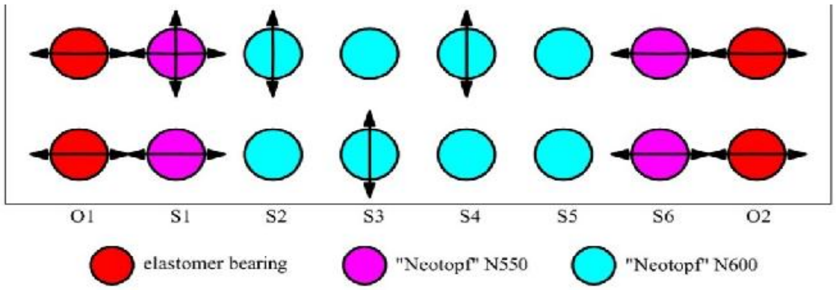
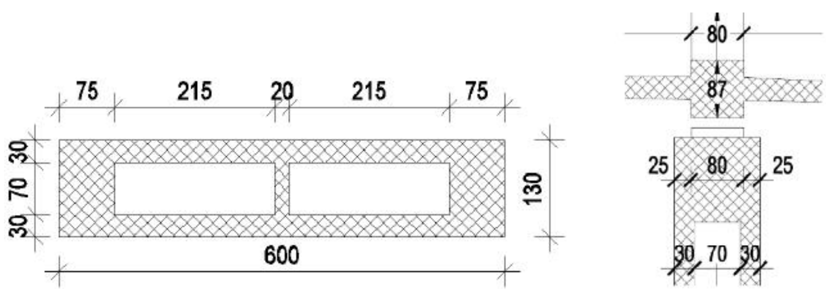


Fig. 3. Bearings layout

The deck structure is supported using different types of "Neotopf" bearings. Some bearings are pinned, some are free to slide in transversal or longitudinal direction. The direction of bearings free horizontal displacements is shown in Figure 3. The longitudinal displacement at the end bearings is limited. The pier S6 has the shear key.

All piers have the dimensions 6.0×1.3m, with 2 inner holes 2.15×0.70m (see Figure 4). The cap beam is positioned at the top of each pier (h_{cb} = 1.30m). The piers are founded on A×B = 4.9x7.6m footings which heights vary from 2 to 10 m. Footings are firmly bonded to the surrounding rock, so they can be considered fixed.



a) *b)*

Fig. 4. Cross section of a) piers and b) pier with the cap beam

3. NUMERICAL MODELS

The considered structure is modelled using FEM. For model generation, calculation and the visualization of results, SAP2000 V14.0.0 software package is used. The numerical analysis is performed using two computational models: beam model and shell model. The shell model has been generated using Bridge Wizard feature of SAP2000 V14.0.0.

For both models, the following calculations have been performed:

a) modal analysis,
b) response spectrum analysis using the horizontal elastic spectrum in two orthogonal directions and vertical elastic spectrum according to EN 1998-1.

3.1. Beam Model

All structural elements have been modelled as isotropic, linear elastic. The material properties of RC members (MB 45 according to Serbian regulations) are E = 34GPa, ν = 0.2, γ = 25 kN/m^3.

All piers have the same cross section, as shown in Figure 4. In the model, the pier lengths are measured from the top of the foundation to the centroid of the cross section of the superstructure: S1=19.02m, S2=19.39m, S3=18.70m, S4=17.99m, S5=17.95m, S6=11.83m. Local axes of the piers have been rotated according to the geometry of the structure: axis 3 is perpendicular to the tangent line of the road, as shown in Figure 5.

From the geological conditions it is concluded that the soil below the foundations is of the high capacity. Therefore, the piers have been modelled as beams completely fixed at the bottom. The moments M3 (axis 3) are free. The cap beams have not been modelled, because their influence on the dynamic behaviour is negligible.

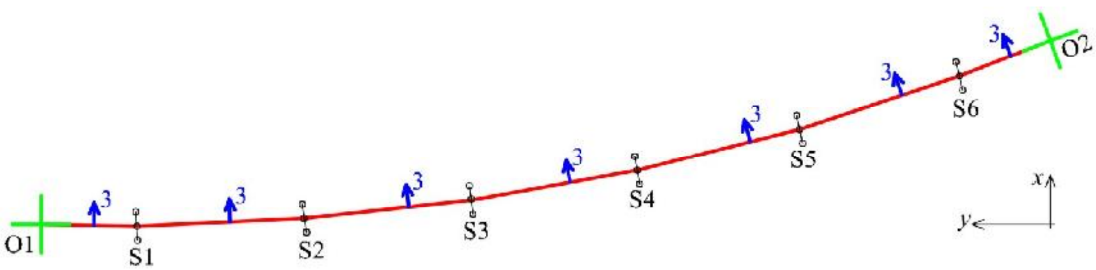


Fig. 5. Bridge plan with local axes of the piers (blue – axis 3, green - abutments)

All longitudinal and lateral variations of the structure geometry have been neglected, because their influence on the global dynamic response is marginal. The cross section of the superstructure is assumed uniform along the entire length. Longitudinal and lateral inclinations have been neglected, as well. The simplified cross section is shown in Figure 6.

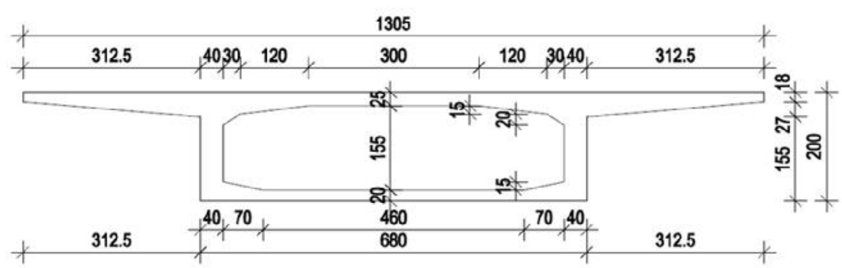
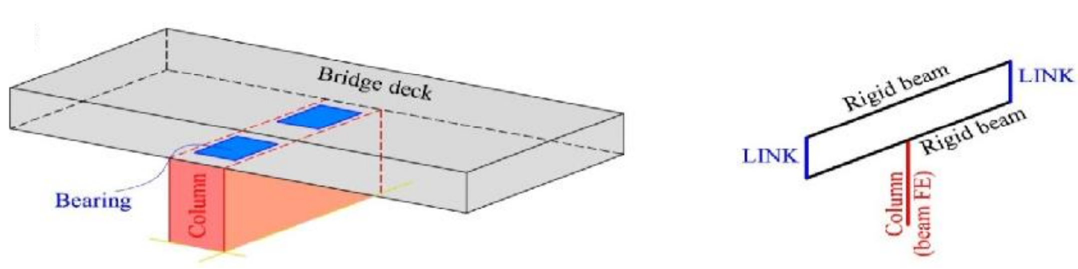


Fig. 6. Simplified cross section of the bridge deck

There are 4 different types of bearings, which is presented in Figure 3. The superstructure is supported using 2 bearings per pier (Figure 7a). The distance between the bearings in the lateral direction is 4.7m. In the numerical model additional rigid beam elements have to be added at the top of the piers in order to introduce different types of bearings (Figure 7b). After that, link element, which stiffness corresponds to the real behaviour of the bearing, is defined in (Ličina and Kovrlija) and ref.5. The rotational stiffness of all bearings is equal zero, while the axial stiffnesses are given in Table 1. Local axes of the bearings are rotated taking into account the real geometry of the deck.



a) *b)*

Fig. 7. Modelling of the supports: a) real structure, b) beam model with link elements

	K1 (vertical)	K2 (longitudinal)	K3 (lateral)
link element 1	916190 kN/m	2322 kN/m	Inf
link element 2	Inf	zero	Inf
link element 3	Inf	zero	Inf
link element 4	Inf	zero	zero

Table 1. Stiffnesses of the link elements to simulate the bearings

Mass of the structure is generated from the self-weight and traffic loading, according to EN 1998- 2. The weights of the structural elements are extracted from the Main Design of Nišava Bridge (Ličina and Kovrlija, 2010). The total self-weight of the superstructure is Q_G = 50498 kN. The traffic loading is accounted as the 20% of the maximum traffic loading according to EC-8-2, and its total amount is Q_S = 2688 kN.

Therefore, the total mass of the superstructure for the seismic calculation is:

$$Q = Q_S + Q_G = 53166\text{kN} \tag{1}$$

The part of the mass has been generated automatically using the options of the SAP2000 from the self-weight of the elements. The additional mass has been added as the line mass.

The model is discretized using beam elements, having the maximum length of 0.5m.

3.2. Shell Model in Bridge Wizard of SAP2000

In contrary to the beam model, the curved axis of the bridge is assigned here using the Layout Lines option of SAP2000. The real curvature is modelled, having the length L=232.2m and radius R=540m. The cross section with its real geometry is modelled using the Deck sections feature, while the bearings are modelled with their real stiffnesses using the single link elements over each pier. The diaphragms with their real thickness have been modelled using the feature of the software. The cap beams have been assigned as beam elements (1.3×1.3m). In this model, the manual rotation of the pier axes is not necessary, because the software performs this operation automatically from the geometry data of the curved superstructure. The longitudinal change of the cross section is modelled using the Parametric Variations feature of the software.

The part of the mass has been generated automatically using the options of the SAP2000 from the self-weight of the elements. The additional mass has been added as the area mass.

Shell elements of the approximate size of 0.5m have been selected for the discretization.

4. MULTIMODAL SPECTRAL ANALYSIS

For the horizontal component of the seismic action, elastic response spectrum, $S_e(T)$, Type 2 for soil type A, according to EN1998-1 is selected, with following parameters corresponding to the spectrum: S=1.00, T_B=0.05s, T_C=0.25s and T_D=1.20s. Characteristic acceleration a_g=0.1g is adopted (VII seismic zone according to Serbian regulations).

Vertical component of the seismic action is modelled by using the vertical elastic response spectrum $S_{Ve}(T)$ according to EN 1998-1, with following parameters corresponding to the spectrum Type 2 and soil type A: a_{Vg}/a_g=0.45, T_B=0.05s, T_C=0.15s, T_D=0.10s.

Both response spectra are presented in Figure 8, according to EN1998-1.

The seismic loading in different directions is generated in SAP2000. The results of the modal analysis of the structure is used to calculate the modal mass ratios (more than 90% of the total mass in the considered direction must be taken into account), while the above described response spectra is used to obtain the appropriate spectral acceleration in the considered mode.

Horizontal spectrum is assigned in two orthogonal directions: lateral (**EQ1**) in the tangential direction in the middle of the bridge tendon (7.5° from the global x-axis, see Figure 5), and longitudinal (**EQ2**), perpendicular to the **EQ1** (97.5° from the global *x*-axis). For the combination of modes, SRSS rule is selected (Clough and Penzien, 1995). Finally, combinations of seismic actions in three orthogonal directions are calculated using the following formulae (EN1998-1) (where **EQ3** is the vertical seismic action):

$$
\begin{aligned}
K_1 &= 1.0\times EQ1 + 0.3\times EQ2 + 0.3\times EQ3 \\
K_2 &= 0.3\times EQ1 + 1.0\times EQ2 + 0.3\times EQ3 \qquad (2) \\
K_3 &= 0.3\times EQ1 + 0.3\times EQ2 + 1.0\times EQ3
\end{aligned}
$$

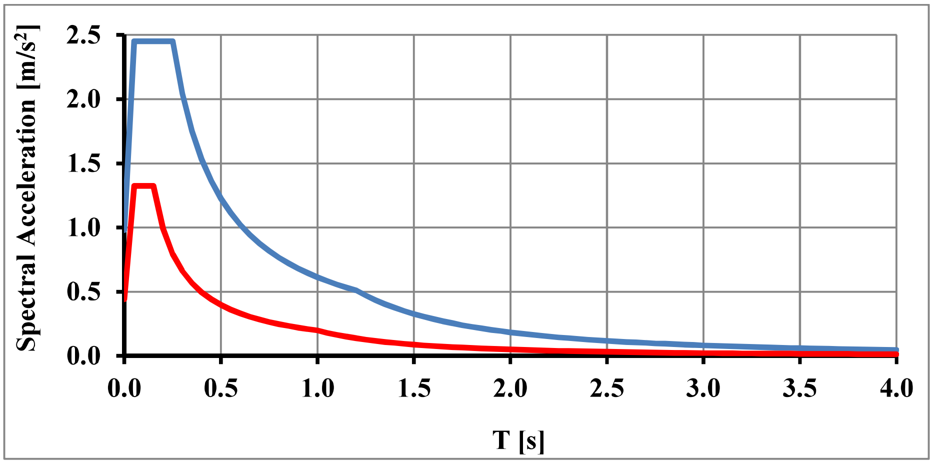


Fig. 8. Horizontal (blue) and vertical (red) elastic response spectrum

5. RESULTS AND DISCUSSION

5.1 Modal analysis

In order to obtained accurate response of the structure the sufficient number of modes shall be taken into account according to EN1998-1. This request is considered fulfilled if one of the following requirements is demonstrated (EN1998-1): (i) the sum of the effective modal masses for the modes taken into account is at least 90% of the total mass of the structure and, (ii) all modes with effective modal masses greater than 5% of the total mass are taken into account. So, in the performed modal analysis, the sufficient number of modes is calculated in both numerical models.

For the comparison, the first three periods and first three mode shapes are provided for both models in Table 2 and Figures 9-11.

Mode	Beam model		Shell model	
1	2.061	longitudinal	2.318	longitudinal
2	0.599	vertical	0.631	lateral
3	0.509	lateral	0.550	vertical

Table 2. First three periods of vibration [s] in both numerical models

From Figures 9-11 it is obvious that there is good match between the mode shapes in two considered models. Beside these modes, there exist a considerable number of "local" modes which influence on the overall response is negligible because of the very low value of effective modal mass (such as local oscillation of a single pier, etc.).

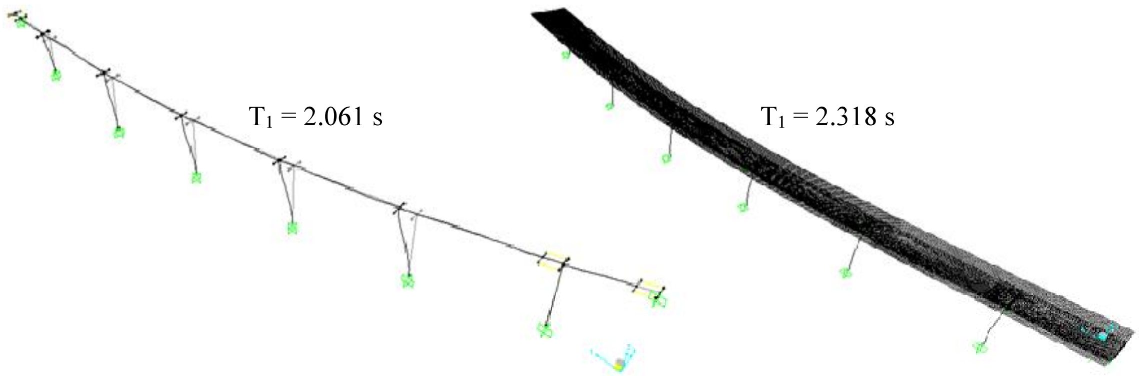


Fig. 9. First mode shape of the RC bridge (longitudinal); left: beam model, right: shell model

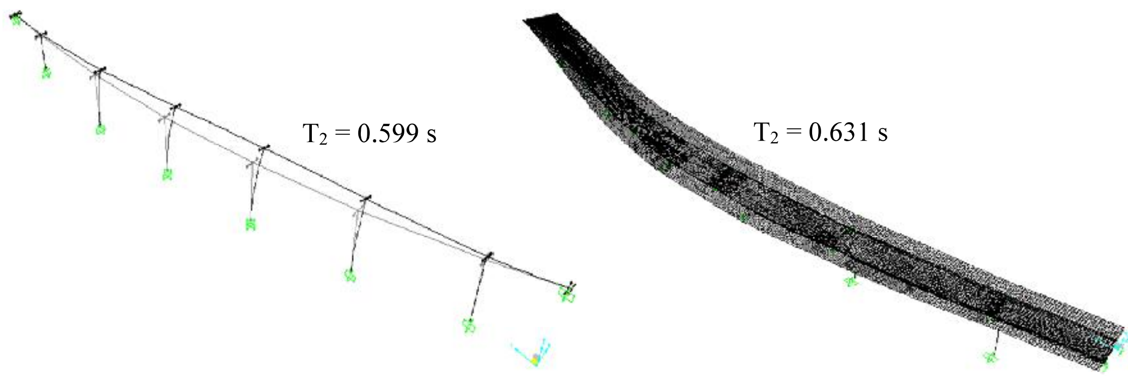


Fig. 10. Second mode shape of the RC bridge (lateral); left: beam model, right: shell model

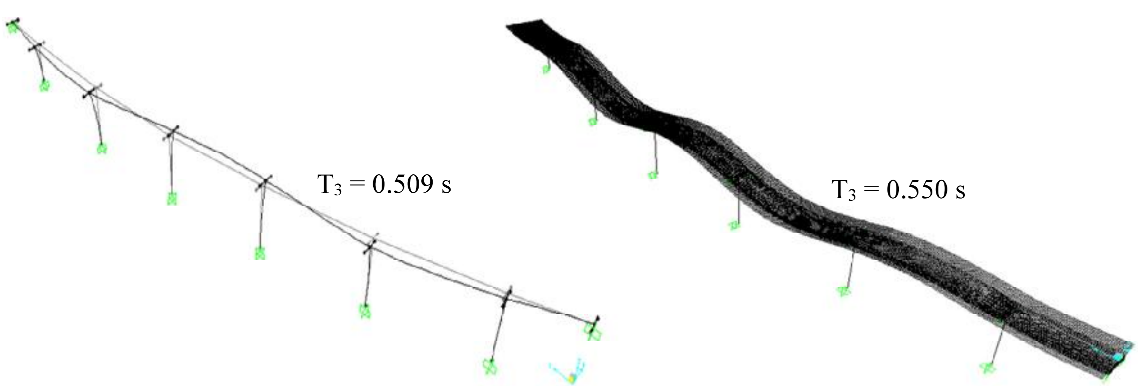


Fig. 11. Third mode shape of the RC bridge (vertical); left: beam model, right: shell model

However, there is a difference in corresponding periods of vibration. Obviously, the beam model is "stiffer" in both longitudinal and lateral direction when compared to the shell model. The difference between periods for the longitudinal direction is 12.47%, while the difference between periods for the lateral direction is 23.97%, which was expected due to the simplifications associated with the modelling of stiffness and mass distribution in the beam model. The periods of oscillation in the vertical direction are in better agreement, where the maximal difference is 8.18%.

There are several sources of differences in obtained periods of vibration: (i) in the shell model, the superstructure is modelled as a curved girder, while in the beam model the polygonal line, from pier to pier, is used; (ii) in both models, the total amount of mass is the same, but in the shell model the additional mass is assigned as an area mass on the top of the plate, having an eccentricity with respect to the centroid of the cross section. On the contrary, in the beam model, the additional line mass has been assigned in the centroid of the cross section of the bridge deck; (iii) the shell model takes into account the longitudinal change of the stiffness of the bridge deck, while the beam model uses the simplified cross section which is constant along the bridge line.

The number of modes required for the accurate calculation of member forces is provided in Table 3 together with the number of modes with M_i>5%:

Direction	Beam model		Shell model	
	Required number of modes	Number of modes with $M_i > 5\%$	Required number of modes	Number of modes with $M_i > 5\%$
EQ1 (lateral)	22	2	51	2
EQ2 (longitudinal)	1	1	1	1
EQ3 (vertical)	200	4	250	3

Table 3. First three periods of oscillation [s] in both numerical models

Shell model requires higher number of modes to be taken into account, because of the greater number of elements, which lead to the more complex approximation of structural mass and stiffness in comparison with the simplified beam model. In both models, the accurate calculation of the structural response in the longitudinal direction requires only the first mode of oscillation. For vertical oscillations, the highest number of modes is taken into account, in both models. However, there are only 1-3 modes with the effective modal mass higher than 5%, depending on the direction of the seismic action.

5.2 Multimodal spectral analysis

After the calculation of modal properties of the structure, the member forces, due to the EQ1, EQ2 and EQ3, are calculated using the multimodal spectral analysis and SRSS method (Clough and Penzien, 1995). The piers member forces are presented in Figures 12-14 (note that only the significant forces have been presented, for the sake of simplicity).

It is obvious that the most important influence which occurs in the piers is the bending moment due to the lateral seismic action EQ1. The bending moments M3 in the lateral directions, due to the seismic action EQ1, are higher for the shell model in comparison with the beam model (Figure 12). The differences are varying from pier to pier (the average difference for all piers is 17 %). Under the seismic action EQ2, the bending moments in two orthogonal directions occur, but their values are approximately 10 times lower in comparison with the M3 values due to the EQ1. Finally, under the vertical seismic action, the normal forces in the piers are the dominant influence. The values of normal forces under the seismic action EQ3 are marginal in comparison with the forces obtained from the self-weight and traffic loading.

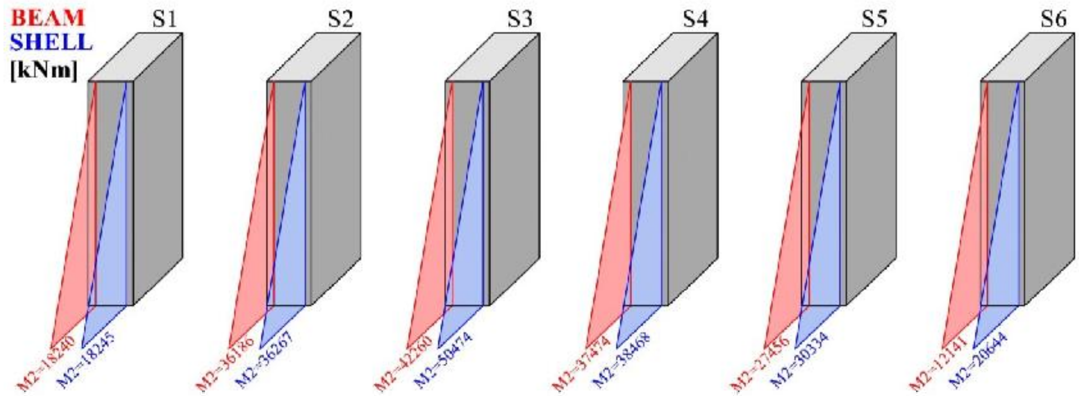


Fig. 12. Force diagrams in piers due to EQ1 (lateral) (N and M3 are not shown)
for two considered models

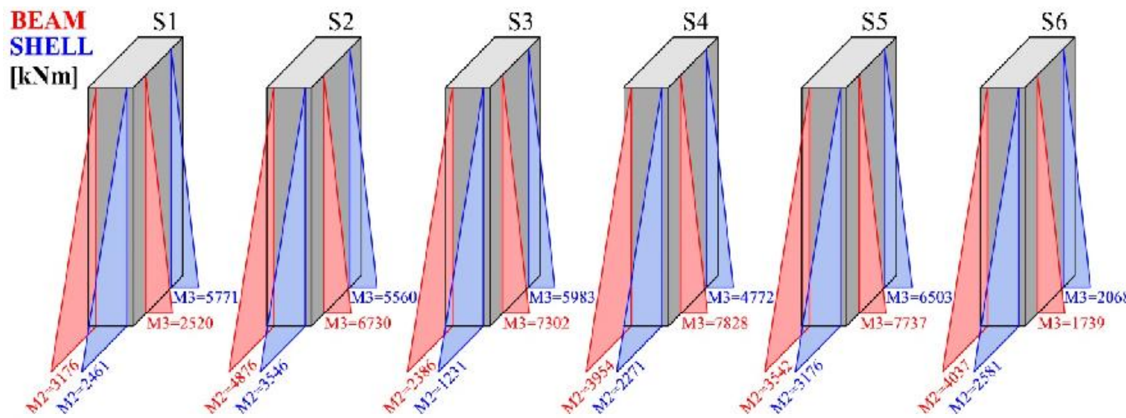


Fig. 13. Force diagrams in piers due to EQ2 (longitudinal) (N are not shown)
for two considered models

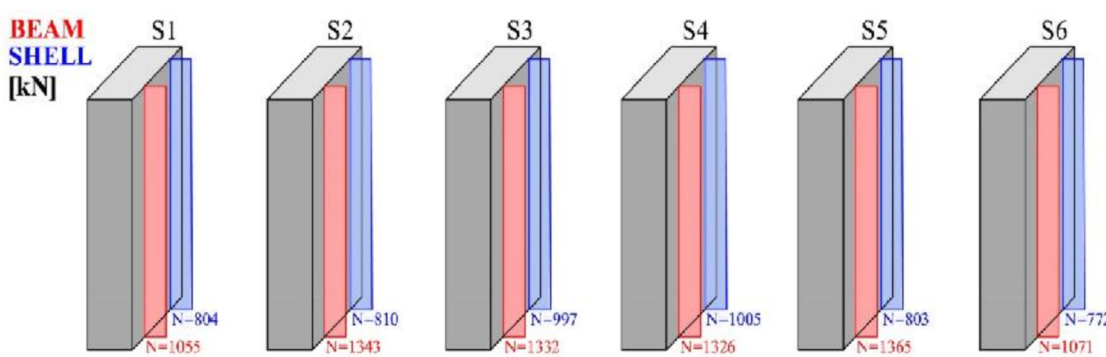


Fig. 14. Force diagrams in piers due to EQ3 (vertical) (M2 and M3 are not shown)
for two considered models

From the above, it can be concluded that the most important influence is the seismic action in lateral direction, EQ1, and the values of the obtained bending moments calculated from two different numerical models are in the acceptable agreement having in mind all simplifications of the beam model in comparison with the real structure.

6. CONCLUSIONS

Two different numerical FE models, beam and shell, of the curved RC bridge structure generated in SAP2000 are presented in this paper. Real, existing structure of the RC bridge on the river Nišava served as an illustrative example. Seismic analysis using multimode response spectrum method according to EN1998-1 and -2 is performed. Some conclusions are elaborated as follows:

- A certain simplifications of the structure are imposed using the beam model, while the shell model (generated by Bridge Wizard feature in SAP2000) represents the structure more realistically. The shell model takes into account longitudinal change of the stiffness of the bridge deck, while the beam model uses the simplified cross section which is constant along the bridge line.
- In the shell model, the superstructure has been modelled as a curved girder, while in the beam model the polygonal line has been used. The total mass is the same, but in the shell model the additional mass has been assigned as area mass on the top plate, while in the beam model the additional line mass has been assigned in the centroid of the deck cross section.
- There is a good matching between the mode shapes in two considered models, with some differences in obtained periods of vibration. The beam model is "stiffer" in both longitudinal and lateral directions due to the simplifications associated with the modelling of stiffness and mass distribution.
- The periods of vibration in the vertical direction are in very good agreement.
- The number of modes required for the accurate calculation of member forces depends on the direction of the seismic action, varying from only the single mode in the longitudinal direction to more than 200 modes in the vertical direction (see Table 3 for details). The shell model requires higher number of modes to be taken into account.
- There are only 1-3 modes with the effective modal mass higher than 5%, depending on the direction of the seismic action.
- The lateral seismic action EQ1 generates the highest stress resultants in the piers. The bending moments M3 are the most important influences, and they are on average 17% higher in the shell model in comparison with the beam one. Under the seismic action EQ2, the bending moments are approximately 10 times lower in comparison with the M3 values under the EQ1.
- The vertical seismic action generates the normal forces in the piers, which are still marginal in comparison with the normal forces in the piers obtained from the self-weight and traffic loading.
- The beam model gives accurate results, and due to its simplicity it can be recommended for dynamic analysis of bridges without any doubt concerning the expected result.

ACKNOWLEDGMENTS

The financial support of the Ministry of Education, Science and Technological Development, Republic of Serbia, through the projects TR-36043 and TR-36048, is acknowledged.

REFERENCES

[1] Computers and Structures, Inc. (1998) “SAP2000: Integrated Finite Element Analysis and Design of Structures”, Berkeley, California, USA.

[2] Chen, Wai-Fah, and Duan, L. (2003) *Bridge engineering - seismic design*, CRC Press, Boca Raton, Florida. USA.

[3] European Committee for Standardisation (CEN), (2006) “ Eurocode 8 - Design of structures for earthquake resistance - Part 2: Bridges “. Brussels, Belgium.

[4] European Committee for Standardisation (CEN), (2006) “ Eurocode 8 - Design of structures for earthquake resistance - Part 1: General rules, seismic actions and rules for buildings”. Brussels, Belgium.

[5] Catalogue of Bearings (in Serbian). GP Mostogradnja, Beograd, Serbia.

[6] Ličina, Ž, and Kovrlija, Z. (2010). “Main Design of Nišava Bridge (in Serbian)”, Saobraćajni Institut CIP, Beograd, Serbia

[7] Priestley, M. J. N., Seible, F., and Calvi, G. M. (1996). *Seismic design and retrofit of bridges*, Wiley, New York, USA.

[8] Clough, R.W, Penzien J. (1995). *Dynamic of Structures*, Computers & Structures Inc, Berkeley, USA.