Valentyne Shynder, Volodymyr Volotsiuga

PRINCIPLES OF MATHEMATICAL MODELING OF STRESS-STRAIN STATE IN REINFORCED CONCRETE BRIDGE SUPERSTRUCTURES WITH THE USE OF MODERN PACKAGES OF FINITE ELEMENT ANALYSIS

1. Finite element method in the study of span structures

A considerable part of highway bridges and overpasses with span structures out of reinforced concrete T- and double-T-section without diaphragms with average and high-tensile reinforcement were built more than 40-50 years ago. Many buildings are obsolete, as they were designed according to the effective regulations of the past times that required loading different from the currently regulated one.

The fast growth of highway transportations, long operational cycles, unfavorable meteorological conditions as well as inadequate technical level of maintenance lead to the development of various defects and damages. Bridge span structures are subjected to the highest level of depreciation since they are significantly loaded plate beam systems.

The modern theory of calculation of bridge concrete structures should use the carrying capacity reserves in a reasonable way, since they are related to the limited development of plastic strains and other forms of physical or geometric nonlinearity of materials in bridge structures.

Thus it is a topical problem to build a more precise nonlinear strain calculation of plate-beam span structures, which would enable with a given field of damage to analyze strength reserves by taking into account the full diagrams of work material strain.

The stated problem is highly non-linear, both physically and geometrically, as a result at the core of its solution method is a numerical method such as finite element method (FEM) which is the most versatile and powerful regarding its possibilities to describe geometric parameters of the problem [1, 2].

Finite element method (FEM) is an efficient numerical method for solving engineering problems. It can be applied to all classes of field distribution problems, including analysis of structures, heat transfer, liquid flow and electromagnetism, because of its ability to use elements of different shapes and sizes in order to gain

grid partitions of any irregular areas, considering that loading and boundary data may be arbitrary. Correlation between FEM and some other methods of engineering analysis is shown in Figure 1 [3].

```
                    ┌─────────────────────┐
                    │ ENGINEERING ANALYSIS │
                    └─────────────────────┘

┌──────────────────┐   ┌──────────────────┐   ┌─────────────────┐
│ CLASSICAL METHODS │   │ NUMERICAL METHODS │─│ ENERGY METHODS  │
└──────────────────┘   └──────────────────┘   └─────────────────┘
  ┌──────────────────┐            ┌──────────────────────────┐
  │ ACCURATE METHODS │            │ BOUNDARY ELEMENTS METHOD │
  └──────────────────┘            └──────────────────────────┘
  ┌──────────────────────┐     ┌─────────────────────────┐
  │ APPROXIMATE METHODS  │     │ FINITE DIFFERENCE METHOD │
  └──────────────────────┘     └─────────────────────────┘
                        ┌──────────────────────┐
                        │ FINITE ELEMENT METHOD │
                        └──────────────────────┘
```
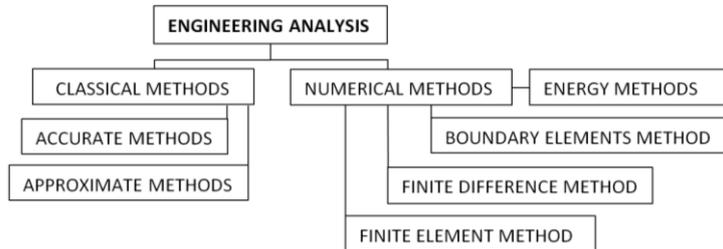
Fig. 1. Correlation between FEM and other methods

FEM is a numerical method for solving differential equations. Therefore it is also a method of building a mathematical model and method of its study.

Nowadays FEM is a universal tool for analysis of structures, which has taken the lead, gradually replacing other numerical methods. Software packages that implement this method play an essential role among numerous CAD/CAM/CAE software programs. The most prevalent packages of this type include Ansys, Nastran, Marc, Fem Models, Impact, ScadSoft, CosmosWorks, Scad, Samcef, Zebulon, Lira, Diana, Rypak, Abaqus, Comsol Multiphysics, Robot and many others.

2. Choosing software for FEM implementation

Femap with integrated solver NX Nastran (NASA STRuctural ANalysis) was selected from among the full range of calculation systems. It is a modern solution from Siemens PLM Software that has proven to be reliable software in complex nonlinear problems solving, it is used by various engineering and technical organizations to solve a large number of diverse engineering and scientific problems. The software system Femap with NX Nastran was chosen as well due to the fact that the staff of the Lviv Polytechnic National University has access to the licensed version [4].

Femap NX Nastran capabilities were analyzed in terms of their use to solve the stated problem, first of all its capability to solve nonlinear problems, fracture mechanics problems, including physical nonlinearity of materials [3], combination of two objects with different characteristics, such as fixture and concrete.

The solver NX Nastran provides the full set of engineering functions [3, 5, 6], such as calculations of strain-stress distribution, natural frequencies and modes of vibrations, stability analysis; solving of heat transfer problems; investigation of steady and transient processes, nonlinear static processes and nonlinear dynamic transients; frequency response analysis and the dynamic and random activity reaction analysis.

Femap with NX Nastran spreads out rapidly in engineering practice thanks to its simplicity, multifunctionality and its ability to function not only on workstations and supercomputers, but also on personal computers. This package runs in Windows environment, so while using it one can access all the features of this operating system. The system provides a high degree of integration with a large number of applications and other systems, such as Abaqus, Algor, Ansys, Caefem, Genesis, Ls-Dyna3D, Marc, Patran, Cosmos and others. Femap supports four types of materials that are used by the Nastran program: isotropic, orthotropic, anisotropic and high-elastic material, allowing to model the full range of constructions.

The works of K.N. Rudakov [5], S.P. Rychkov [3], D.G. Shymkovych [6], T. Dmitrenko [7] and others contribute to the research in Femap-Nastran.

Considering the above-mentioned and reasoning from the fact that the application of systems for modern automated analysis, engineering and calculations gives a number of advantages in practical construction and scientific research performance, the use of such packages should be regarded as relevant and promising.

3. Problem definition

Despite the fact that the NX Nastran system can solve extremely wide range of different kinds of problems, at the moment it is used more in such areas as aerospace, motor-vehicle construction, shipbuilding, machine building, medicine, fast moving consumer goods, providing analysis of strains, vibrations, durability, heat, noise, acoustic transfer and so on. Whereby, the use of the FE analysis system in construction is observed mainly regarding the metal structures, especially in Ukraine. As for reinforced concrete, including bridge structures and systems, their exploring using NX Nastran is only starting to pick up steam. Therefore, the use of advanced package of FE analysis for reinforced concrete bridges is a relevant issue.

4. Peculiarities of reinforced concrete modeling

Analysis of reinforced concrete bridges by FEM in Femap-Nastran NX is supposed to be performed in the following order:
- development of geometric model construction (or use of a ready one by the means of import);
- input of materials characteristics;
- specification of the FE types and their properties;
- creation of grid partition for the FE model;
- check of the FE partition correctness and efficiency;
- input of FE model boundary conditions;
- forming of the load conditions for the FE model;
- check of the model correctness and rationality;
- choice of type and parameters for the FE analysis;
- performance of the FE analysis for the created model;

- analysis of the calculated results, identification of possible errors, if necessary, introduction of alterations and subsequent FE analysis;
- processing, storage and presentation of calculation results.

At the preparatory stage of calculation model creation, attention should be paid to:
- possibility to describe the specific construction with those types of boundary problems, which the NX Nastran can solve;
- previously to review available types of finite elements (FE) (since each FE type has its own properties) and determination of whether they are appropriate to model the problem given;
- thickening the FE grid in areas of expected stress buildup, and in the vicinity of the imposed boundary conditions;
- if there is a geometric plane of symmetry, and if therewith loading conditions are as well symmetric to this plane, it is recommended to replace the symmetric part of the symmetry boundary conditions (for example, for problems associated with strains, one can restrict nodal degrees of freedom of finite-element net, which are located in the plane of symmetry in the direction perpendicular to this plane;
- applying of concentrated force to a single grid node will cause significant local stress concentration, thus these forces are recommended to replace with the distributed ones, because these concentrated forces do not exist in reality;
- necessity to have the input data and specifications for materials that need to be entered depending on the accepted model of the material.

FEM calculations of the reinforced concrete have its own specific peculiarities. In the case of preliminary reinforcement stressing it is important to calculate correctly all the prestressing losses in steel rebar, and also in concrete under compression, including time-dependent deformation. Consideration of the physical non-linearity and secondary effects requires more resources, especially those of the machine. The difficulty of concrete modeling with Femap is also a need to ensure its exclusion from functioning in the tensile region while maintaining work in the transverse direction. This is implemented by considering the anisotropic properties of the material. The result of the calculations depends largely on established analytic model.

Particular attention should be paid to the creation of the finite element grid. Structures like reinforced concrete where steel rebar is encased in solid concrete can be represented by 1-D rod or beam elements for the rebar inside and solid 3-D tet-elements for the concrete. When setting up such a model, we have to take care to ensure that the mesh is contiguous and that the 1-D elements connect with the 3-D elements correctly.

By default, when you enter the characteristics of the new material, the material is considered to be isotropic. The type of material can be chosen from the following: Isotropic; Orthotropic (2-D); Orthotropic (3-D); Anisotropic (2-D); Anisotropic (3-D); Hyperelastic; Fluid. Input of the characteristics of the material is per-

formed in the pre-selected system of units. After entering the characteristics of the material the model of calculation measurement system of units becomes completely defined and invariable.

For the isotropic material it is necessary to enter two of three following parameters: Young modulus, E (E $\geq$ 0); Shear modulus, G (G $\geq$ 0); Poisson's ratio, ν (1 $< \nu \leq$ 0.5). If one of these parameters is not specified, it is calculated from the equation:

$$G = \frac{E}{2(1+\nu)}$$

if all three parameters are given, it is recommended for them to be in the following correlation:

$$\left| 1 - \frac{E}{2(1+\nu)} \right| < 0.01$$

If not to specify the necessary data for the defined type of problem, such as Young's modulus and/or Poisson's ratio (linear elastic problem), yield stress limit «Initial Yield Stress» (plastic material), the density of the material (for non-stationary heat conduction and dynamic problems), a critical error can occur when trying to analyze the corresponding problem.

5. Static elastic calculation of reinforced concrete bridge beam

Let us consider the main reinforced concrete beam of the bridge test item with the length of 25.56 m (bearing distance is 25 m). The longitudinal girder crosswise beam in the process of installing is being combined into a dimensional system of a bridge superstructure (span structure) on the level of shelf beams with the width of 15 cm with the embedded longitudinal joint with the width of 0.35 m with loop protruding bars. The width of the shelf of the main beam under consideration is 1.0 m, the height is 1.2 m.

For beams it is accepted the concrete of B30 class with the following design characteristics: Design strength of concrete to compression (prism strength) is R_b = 15.5 MPa, design strength of concrete to axial tension is R_{bt} = 1.1 MPa, elastic modulus is $E_b = 32.2 \cdot 10^3$ MPa. The longitudinal effective reinforcement bar is accepted to be of A-III class with the following properties: design strength to stretch and compression is $R_s = R_{sc}$ = 350 MPa, elastic modulus is $E_s = 200 \cdot 10^3$ MPa. Bent-up bars of the A-III class with the design strength of fixture to tension in the oblique section is R_{sw} = 280 MPa.

Reinforcement of the beam and its cross-section is shown in Figures 2 and 3.
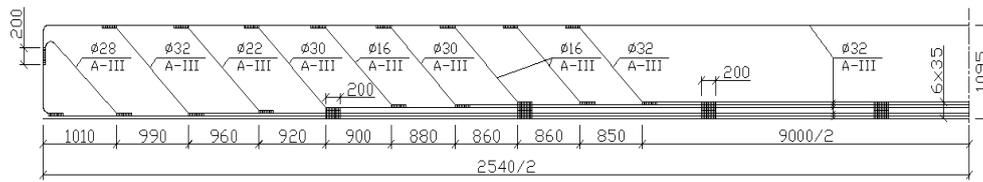
Fig. 2. K-1 framework of the beam reinforcement (2 pcs.)
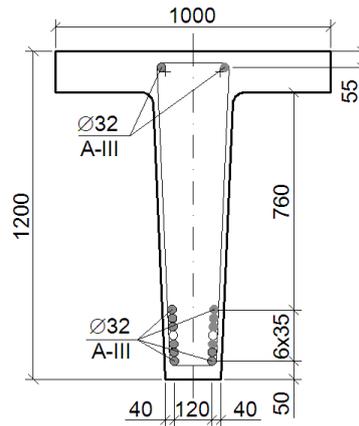
Fig. 3. Cross-section of the beam

Beam rests on two piers, a pivotally movable one and a pivotally immovable one. The distributed load is evenly applied on the top surface of the beam shelf with the value of 37.20 kN/m^2. Finite element model of this beam is shown in Figures 4 and 5.
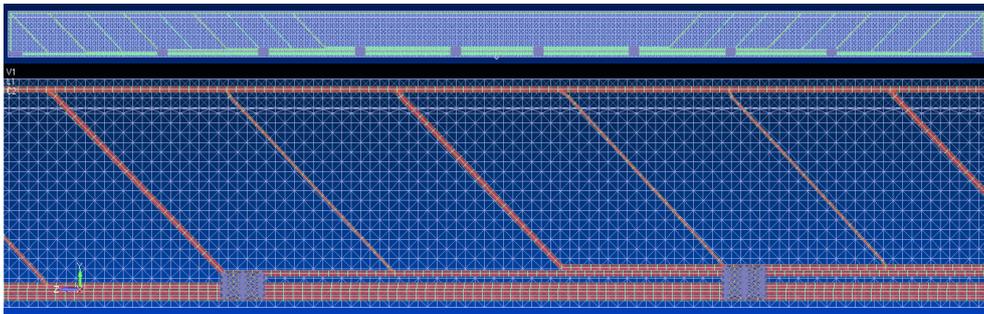
Fig. 4. Top views of FE model

For the given model with the program NX Nastran there was performed a static elastic calculation (1 .. Static). The maximum deflection is 59.5 mm.
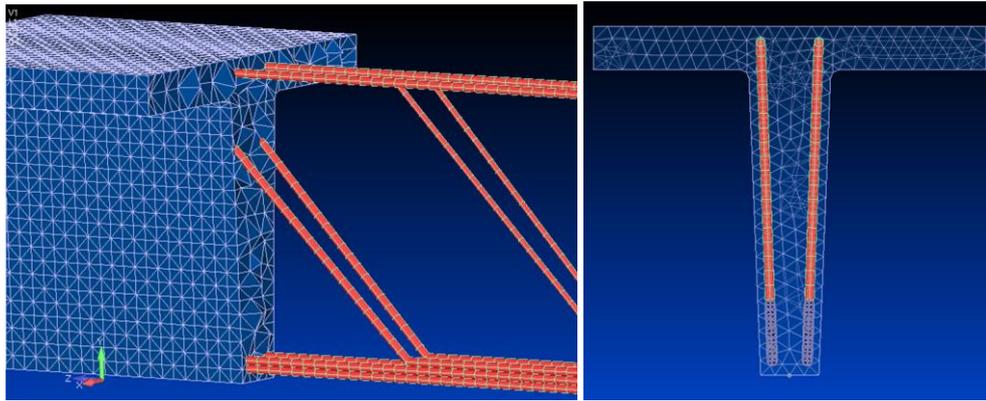
Fig. 5. 3-D views of FE model
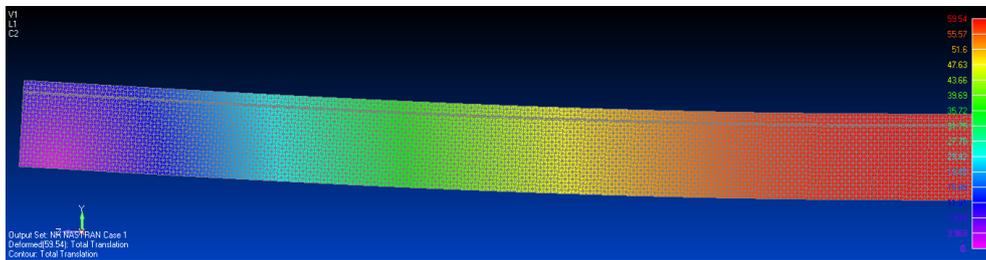
Fig. 6. Total translations
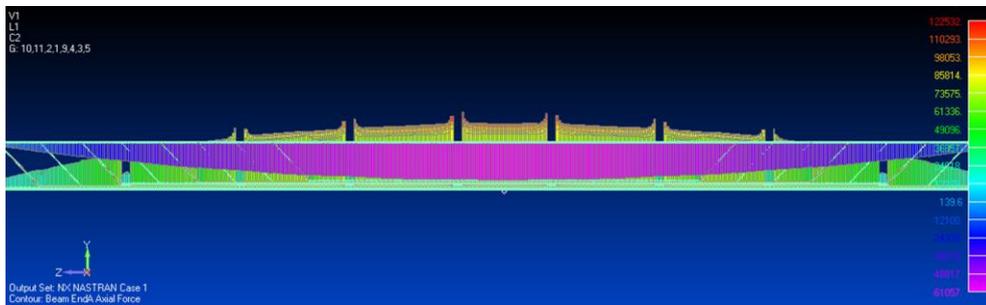
Fig. 7. Beam (steel rebar) axial force

Figure 4 shows a transparent finite-element model of the beam from the side. It can be seen that the stretched zone has six rows of reinforced bars, while in order to meet the standard requirements [8] after every three rows of welded frame openings are designed equal to the diameter of the fixtures.
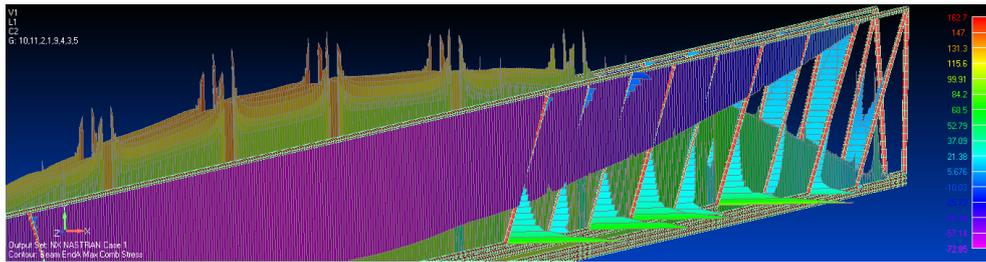
Fig. 8. Beam (steel rebar) MaxCombStress

The openings are formed with the installing of reinforcing corbels with the 20 cm length in the places of bends beginning and along the frame. Exactly due to the welding of the frames in the places of these corbel inserts and consequently to the increased rigidity in areas of welding as the result of analysis on the diagram (Figs. 7 and 8) jumps and stress concentration in these areas can be observed.

6. Consideration of nonlinearity in Femap

The difficulty of building theoretical reinforced concrete models, except for the anisotropy, is caused by the elastic-plastic characteristics of the material concrete itself. Concrete is characterized by non-linear relation between stress and strain, different mechanics under compression and stretch etc. Historically, the lack of sufficient experimental data on the relation "stress-strain" (σ-ε) for different types of concrete led to the use of the law of concrete strain in a simplified, idealized form.

European standards [9] also adopted the diagram «σ-ε» with the flowing sprig: curved and two versions of simpler nonlinear diagrams without flowing sprigs, and with limited yield area (three-linear and bilinear - in the form of Prandtl diagram).

Numerical examination of concrete structures, with respect to the real properties of materials leads to the need to solve nonlinear algebraic equations systems. The nonlinearity of the equations of the problem is primarily caused by the dispropor-tionate link between stress and strain. It is a physical type of nonlinearity. Geometric nonlinearity occurs when the equilibrium equation is written down in the deformed scheme, i.e. movement of structures causes a significant change in its geometry. Regardless of the type of nonlinearity the solving equation for the finite element method is the equation of equilibrium. In this case, all the information about the physical and geometric nonlinearity is contained in the stiffness matrix design, the components of which are connected with the stiffness matrix of indi-vidual elements.

The program Nastran has three models of nonlinear materials implemented: Non-linear Elastic, Elasto-Plastic, Hiperelastic. The model of high-elastic (Hiperelastic) materials is singled out as a separate type of material. Setting of nonlinear elastic material models or of elastoplastic material is performed while determining the

properties of isotropic, orthotropic and anisotropic materials among three options: Nonlinear Elastic; Elasto-Plastic (Bi-Linear); Plastic.

Nonlinear elastic material is defined by the curve "stress-strain", which is set in the model using the functional relation of the strain versus stress (type of relation "..vs. Strain") or stress versus strain (type of relation "..Stress vs. Strain"). This function must be defined in the first and third quadrants of the plane σ-ε and it can describe different properties of the material under axial stress and compression. Loading and unloading of material is performed at the same curve without residual strain. Nonlinear elastic properties can be set only for isotropic materials. For orthotropic and anisotropic materials the "Nonlinear Elastic" material model cannot be set.

While setting bilinear elastic plastic material it is necessary to enter the following data: Plasticity Modulus, H; Hardening Rule (0..Isotropic; 1..Kinematik; 2..Isotropic+Kinematik); Yield Function - Yield Criterion (0..von Mises: 1..Treska; 2..Mohr-Colomb; 3..Drucker-Prager), Initial Yield Stress.

"Plasticity Modulus" for elastic plastic material with piecewise linear approximation (Elasto-Plastic, Bi-Linear) is calculated by the formula:

$$H = E_t/(1 - E_t/E)$$

where E is Young's modulus, E_t is tangent modulus of the second section of the curve uniaxial strain.

Hardening Rule. In the isotropic model (0..Isotropic) the yield limit of compression strain equals the plastic stretching strain. Isotropic model is the simplest and the most commonly used one. For kinematic model of strengthening (1..Kinematik), the value of elastic unloading equals the double initial yield limit. The real strengthening of material takes place in a curve lying between these extreme models of strengthening.

Yield Function. The Mises criterion is the most general criterion for the analysis of plastic behavior of structural materials. The Treska criterion is more efficient in use for the study of brittle materials. The Mohr-Coulomb and the Drucker-Prager criteria are developed for materials with internal friction, such as soil and concrete.

The value of plasticity function, which are calculated according to the Mises and Treska criteria are compared with the initial value plasticity limit, which is set in the field "Initial Yield Stress".

Elastic plastic material of the general form. This material model has no fundamental differences from the previous model, except for that this strain diagram may consist of many linear sections, and therefore it is set differently. Instead of the strength coefficient there is set the functional stress strain relation. Unlike the functions that are generated for nonlinear elastic material, this function must be defined in the first quadrant (σ > 0; ε ≥ 0) and must have the inclination of the first section, which is equal to modulus of elasticity, given in the basic properties of the material input.

Consequently, calculations of problems using the finite element method are becoming more popular, especially due to the rapid development of technologies and their capabilities. It should be noted that these calculations are approximate. To get maximally accurate results the maximum distance between the nodes of the finite-element grid should be reduced and the finite-elements of the second order approximation (parabolic) should be applied; this often leads to a dramatic increase of nodes and finite-elements, which entails the need to increase the RW and disk storage memory and requires more time to solve the system of algebraic equations. In addition, the process of computational model creation is often quite difficult and time consuming. It is a creative process in its own way and the quality together with the approach chosen influence the accuracy of the result, the calculation time and the accuracy (convergence) of results.

Conclusion

In this article has been suggested the use of the software package Femap-Nastran NX for calculations and analysis of reinforced concrete bridges using the finite-elements method. The use of such package of the finite-element analysis has several advantages, among which is ability to create models which are at the most close to the real conditions. Herein it has been shown the algorithm of the finite element analysis of such structures performed in Femap-Nastran NX. Herein there have been reviewed some peculiarities and capabilities to consider the nonlinearity in the software complex Femap NX Nastran.

References

[1] Bazhenov V.A., Perelmuter A.V., Shishov O.V., Budivelna mechanica. Compuyterni techniligyy, [Structural Mechanics. Computer technologies], Karavela, Publ., Kiev 2009, 696 p.

[2] Klovanych S.F., Metod konechnykh elementov v nelynenykh zadachakh inzhenernoi mekhaniki, [The finite elements method in nonlinear mechanical engineering problems], Library journal "The World of Geotechnology" 2009, 9th edition, Z.: IPO ltd., Zaporozhye, 400 p.

[3] Rychkov S.P., Modelirovanie konstruktsii v srede Femap s NX Nastran, [The modeling of structures in the Femap environment with NX Nastran], DMK Press, Publ. Moscow 2013, 784 p.

[4] Volotsiuga V., Shynder Yu., Use of modern finite element analysis packages in calculations of reinforced concrete bridges, Materials of 5th International Academic Conference of Young Scientists "Geodesy, Architecture and Construction 2013", Lviv Polytechnic National University, November 21-23, 2013.

[5] Rudakov K.N., FEMAP 10.2.0. Geometricheskoie i konechno-elementnoie modelirovaniie konstruktsii, [Geometric and finite-element modeling of structures], NTUU "KPI", Publ., Kiev 2011, 318 p.

[6] Shymkovych D.H., Femap & Nastran. Inzhenernyy analiz metodom konechnykh elementov, [Engineering finite element analysis], DMK Press, Publ. Moscow 2008, 704 p.

[7]   Dmitrenko T., Vykorystannia novitnikh kompiuternykh tekhnolohii pry chyselnomu doslidzhenni napruzheno-deformovanoho stanu budivelnykh konstruktsiyi, [Use of the latest computer technology in the numerical study of stress-strained state of constructions], Visnyk L.P., "NU" 2013, 751, 346-350.

[8]   Ukrainian national construction regulation B.2.3-14:2006. Transport facilities. Bridges and pipes. Design rules, Ukraine Ministry of Regional Development, Kyiv 2006, 376 p.

[9]   ENV 1992-1-1: 1991: Eurocode 2: Design of concrete structures - Part 1-1: General rules and rules for buildings, European Prestandart June, 1992, 116 p.

[10] Perelmuter A.V., Slivker V.I., Raschetniie modeli sooruzhenii i vozmozhnost ikh analiza [Computational models of structures and their possible analysis], 4th-ed., DMK Press, Publ. Moscow 2011, 736 p.

Abstract

This article describes the role of the finite element method (FEM) in the study of span structures. The choice of software for the FEM implementation has been rationalized. The use of the modern package of finite element analysis Femap with NX Nastran in order to model and calculate reinforced concrete bridge span structures has been reviewed. The sequence for reinforced concrete analysis has been suggested, and modeling peculiarities together with options to include non-linearity in the program have been described. Herein it has also been performed the static elastic calculation of reinforced concrete beam in Femap-Nastran NX.

Zasady matematycznego modelowania stanu naprężenia i odkształcenia żelbetonowych nadbudówek z wykorzystaniem nowoczesnych pakietów analizy elementów skończonych

Streszczenie

W artykule opisano rolę metody elementów skończonych (MES) w badaniu nadbudówek. Został zracjonalizowany wybór oprogramowania dla realizacji MES. Omówiono zastosowanie nowoczesnego pakietu analizy elementów skończonych Femap z NX Nastran w celu modelowania i obliczania konstrukcji mostowych żelbetowych. Zasugerowano sekwencję analizy zbrojonego betonu i opisano osobliwości modelowania wraz z opcjami w programie uwzględniającymi nieliniowość. Wykonano również obliczenia statyczne elastycznej belki żelbetowej w Femap Nastran NX.