Use of modern finite element analysis packages in calculations of reinforced concrete bridges

Volodymyr Volotsiuga, Yuliya Shynder

Bridges and building mechanics Department, Lviv Polytechnic National University, UKRAINE, Lviv, Karpinskogo street 6/201, E-mail: volotsuha@gmail.com

***Abstract* – This article deals with the use of modern finite element analysis package in calculation of reinforced concrete bridges. The static elastic calculations of reinforced concrete bridge girder was being performed in Femap-Nastran NX.**

Keywords – Finite Element Method (FEM), reinforced concrete bridges, Femap, Nastran NX, finite element (FE) modeling.

I. Introduction

Finite element method (FEM) is an efficient numerical method for engineering problems solving. 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 FME and some other methods of engineering analysis is shown in Fig. 1 [1].

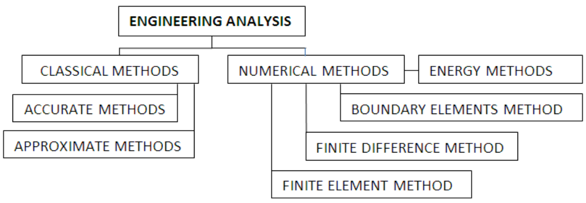

Fig. 1. Correlation between FME and other methods

Nowadays FEM is a universal analysis structure tool. 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.

One of the best modern multi-functional programs implementing FEM is considered to be Femap with the built-in solver NX Nastran (NASA STRuctural ANalysis), which is a modern solution from Siemens PLM Software. The solver NX Nastran provides the full set of engineering functions 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 eingineering 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.

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 perfomance, the use of such packages should be regarded as relevant and promising.

II. 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.

III. Analysis of recent research and publications

Implementation of advanced computer technology in engineering practice requires organization of corresponding courses and disciplines, and it can be observed in all the leading educational and scientific institutions of Ukraine. For example, in 2009, with the order of the Rector of the Lviv Polytechnic National University there was established the training center of automatic calculations of engineering constructions n.a. M.T.Huber [2], which deals with methodical ware and teaching of such subjects as computerized analysis of engineering constructions with the finite element method using the Femap-Nastran program under the guidance of prof. Kharchenko E. V.

Department of Bridges and building mechanics of Lviv Polytechnic National University under the guidance of prof. Kvasha V.H., among other things, engaged in the construction models of nonlinear reinforced concrete bridge systems, development and introduction of modern software in the bridges engineering and reconstruction, in the analysis of strain stress distribution of constructions and so on.

The works of K.N. Rudakov [3], S.P. Rychkov [1], D.G. Shymkovych [5], T. Dmytrenko [6] and others contribute to the research in Femap-Nastran.

IV. The basic material statement

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.

FEM calculations of the reinforced concrete have its own specific pecularities. 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 nonlinearity 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.

Let us consider the main reinforced concrete beam of the bridge test item with the length of 25.56 m. The longitudinal girder crosswise beam in the process of installing is being combined into a dimensional system of a bridge superstructure on the level of shelf beams with the width of 15cm with the embedded longitudinal joint with the width of 0.35 m. For beams is accepted the concrete of B30 class with the calculated characteristics: R_b=15,5 MPa, R_{bt}=1,1 MPa, $E_b=32{,}2\cdot10^3$MPa. The longitudinal effective reinforcement bar is accepted as the one of A-III class with the following properties: $R_s=R_{sc}$=350MPa, $E_s= 200\cdot10^3$MPa. Bent-up bars of the A-III class with R_{sw} = 280MPa. 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 18.20 kN/m2.

For the given model with the program NX Nastran was performed a static elastic calculation (1 .. Static). The maximum deflection is 59.5 mm.

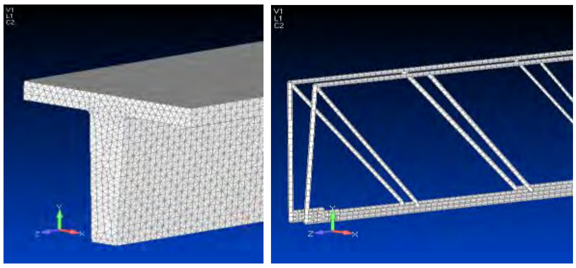

Fig.2 Finite-element model of concrete and steel rebar

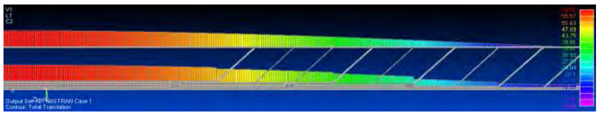

Fig.3 Beam elements displacement diagram

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 FE analysis has several advantages, among which is ability to create models which are at the most close to the real conditions. Herein has been shown the algorithm of the FEM analysis of such structures performed in Femap-Nastran NX. Also there have been conducted and modeled the static elastic calculations of reinforced concrete bridge beam.

References

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

[2] “Training center of automated calculation of engineering constructions n.a. Huber,” lp.edu.ua. [Online]. Available: http://lp.edu.ua/node/2590 [Accessed: Nov. 1, 2013].

[3] K.N. Rudakov. FEMAP 10.2.0. Geometricheskoie i konechno-elementnoie modelirovaniie konstruktsii [Geometric and finite-element modeling of structures]. Kiev: NTUU "KPI" , Publ.2011, pp. 317.

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

[5] T. Dmitrenko “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”], Lviv, Visnyk LP “NU” No.751, pp 346-350, 2013.

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