

FACTA UNIVERSITATIS

Series: **Architecture and Civil Engineering** Vol. 8, № 2, 2010, pp. 201 - 210

DOI: 10.2298/FUACE1002201B

**MATHEMATICAL MODELING OF MATERIALLY NONLINEAR
PROBLEMS IN STRUCTURAL ANALYSES
(PART II – APPLICATION IN CONTEMPORARY SOFTWARE)**

UDC 624.01:519.711(045)=111

**Zoran Bonić, Todor Vacev, Verka Prolović,
Marina Mijalković, Petar Dančević**

University of Niš, Faculty of Civil Engineering and Architecture, Serbia

Abstract. *The paper presents application of nonlinear material models in the software package Ansys. The development of the model theory is presented in the paper of the mathematical modeling of material nonlinear problems in structural analysis (part I - theoretical foundations), and here is described incremental-iterative procedure for solving problems of nonlinear material used by this package and an example of modeling of spread footing by using Bilinear -kinematics and Drucker-Prager mode was given. A comparative analysis of the results obtained by these modeling and experimental research of the author was made. Occurrence of the load level that corresponds to plastic deformation was noted, development of deformations with increasing load, as well as the distribution of dilatation in the footing was observed. Comparison of calculated and measured values of reinforcement dilatation shows their very good agreement.*

Key words: material models, nonlinearity, Bilinear kinematic, Drucker Prager, experiment.

1. INTRODUCTION

Finite element methods (FEM) is now widely applied in engineering practice through a number of modern software providing large, almost unlimited possibilities to solve the problems of structural analysis. A large family of the software can be globally divided into two groups: software for general purpose and specialized software.

By using a general-purpose software (ANSYS, NASTRAN, ABACUS, DIANA, PAK) high-quality and realistic simulation of building structures can be obtained, and the results are static and deformation parameters. The output results are then subjected to engineering judgment with the empirical data of the materials, service specifications and so on.

Received June, 2010

By using specialized software in domain of civil engineering (SAP, STAAD, TOWER, STRESS) structures are also modeled (with much higher approximations) and basic output parameters are transferred to the appropriate algorithms for the dimensioning of structural elements according to the desired codes. The fundamental disadvantage of the software is that it cannot describe the materially nonlinear problems, but remain in the linear theory of elasticity.

Therefore, the specialized software enhance the quantitative performance of a designer during analysis, while general purpose software change the designer's way of thinking in qualitative terms, freeing him from compromises and limitations.

In order to solve problems treated in this paper, where material nonlinearity, different materials and space state of stress and strain are present, the usage of general-purpose software is the logical choice. For the analysis of examples presented in this paper the ANSYS software has been applied.

2. MODELING OF MATERIALLY NONLINEAR PROBLEMS IN ANSYS

ANSYS allows the application of different treatment of the plastic behavior of materials by applying more rheological models, given in Table 1.

Table 1 Summary of Plasticity Options in ANSYS [1]

Name	Yield Criterion	Flow Rule	Hardening Rule	Material Response
Bilinear Isotropic Hardening	von Mises/Hill	associative	work hardening	bilinear
Bilinear Kinematic Hardening	von Mises/Hill	associative	kinematic hardening	bilinear
Multilinear Isotropic Hardening	von Mises/Hill	associative	work hardening	multilinear
Nonlinear Isotropic Hardening	von Mises/Hill	associative (Prandtl-Reuss equations)	kinematic hardening	bilinear
Multilinear Kinematic Hardening	von Mises/Hill	associative	kinematic hardening	multilinear
Nonlinear Kinematic Hardening	von Mises/Hill	associative	kinematic hardening	nonlinear
Anisotropic	modified von Mises	associative	work hardening	bilinear, each direction (tens. and comp.) different
Drucker-Prager	Mises-dependence on hydrostat.stress	associative or non-associative	non	elastic-perfectly plastic

In the linear structural analysis the displacement vector at which the equilibrium of external and internal forces is established can be determined directly - by solving the corresponding system of equations. In the nonlinear finite element analysis, which is used in ANSYS, the relationship between force vector and displacement vector is not linear so solution of the problem is discretization in space (a network of finite elements) and time (time increments).

The essence of incremental procedure for solving non-linear material problems is that the nonlinear behavior approximates linear in small steps, and in each step the linear material behavior law is applied. In order to achieve equilibrium during each increment the iterative procedures are used, so the combination of these two procedures is called the incremental-iterative procedure. In order to solve nonlinear problems Ansys uses Newton-Raphson's process, where the total load is divided into a series of incremental loads, and each of them into several load steps.

Before each iteration, the Newton-Raphson's method to calculate a vector of unbalanced load is used, which represents the difference between load corresponding to stress in element and the applied load. Then, the program implements linear resolution using unbalanced load and tests the convergence of solution. Unless the convergence criterion is satisfied, the unbalanced load vector is re-determined, stiffness matrix is recalculated and the new solution is obtained. This iterative procedure continues until solution converge. Implementation of described incremental-iterative procedure in ANSYS is presented in detail in [1].

3. DESCRIPTION OF EXPERIMENTAL RESEARCH

To illustrate the application of the explained methodology of calculation the example of experimentally investigated spread footings of the project of technological development TR16021 funded by the Ministry of Science and Technology Technological Research of the Republic of Serbia is used. The goal of research is to determine, theoretically and experimentally, behavior of spread footings of reinforced concrete supported by a deformable subsoil and loaded by external load till failure. In the course of the process it was important to determine distribution of stresses and strains in the reinforcement and concrete of the footings, as well as the influence of the factors such as: compressive strength of concrete, applied reinforcement percentage, position and diameter of reinforcement, type and characteristics of bed and uniformity of contact pressures etc. The key result of conducted research should be determination of those factors whose influence is dominant in foundation design.

The program of experimental research in the Project included production of a model *in situ* with preparation of the subsoil of proscribed geomechanical characteristics and production of test elements – spread footings of given dimensions and defined characteristics of concrete and reinforcement. Figure 1 provides scheme of the experiment. The following measuring equipment was provided for the measuring in the subsoil and footings: measuring dilatation strips to measure dilatation in concrete and reinforcement of the foundations, for registering contact pressures – measuring pressure cells, for registering foundation displacement - instrument LVDT as well as hydraulic jack with 1000kN capacity and force sensor with the same capacity.

As may be seen from the Figure 1, the experiment required construction of a steel frame intended for reception of reactive force of the press used to load foundations. The frame is a steel space structure consisting of the bottom and space frame (a pair of vertical members, two pairs of struts and transversal beam). Mutual joints of the elements are realized by bolts. The following steel elements and sheet metal were used for the structure: transversal beam - IPB HE-B600, vertical members - U200, struts - U200, bottom – sheet metal d=5 mm, bottom frame L100x10. Apart from the conditions of stress and strain, the frame had to be cost efficient and

simple for manufacturing and assembling in the field. That is why the number of elements is minimal and joints secured by bolts through nodal sheet metal. More details on the steel frame is given in [7].

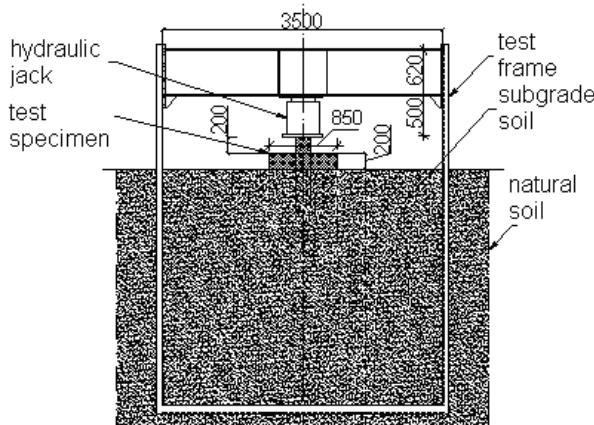


Fig. 1 Scheme of the experiment

The adopted dimensions of the test elements – spread footings were 85x85cm at the base which is approximately 1/3÷1/2 of the footings dimensions regularly used in building construction. The height of the footings ranges between 12.5cm to 25cm and the concrete grade from MB15 to MB25. Diameter of applied reinforcement is Ø 8, 10 and 12mm and reinforcement percentage approximately 0.4%. The characteristics of the footings were adopted on the basis of characteristics of the footings tested in earlier experiments.

Also the form of the diagram of contact pressures for various rigidity of the system foundation-subsoil was monitored, as well as the discrepancy between parameters of average contact pressures beneath the entire foundation and average contact pressures underneath the punching cone. This discrepancy is particularly interesting considering that in the punching shear calculation, value of average strength under the entire foundations is used, instead of real average value of contact pressures underneath the punching cone.

Experiment is performed at the location of the asphalt plant "Niskogradnja" A.D. in Niš, participating in the project, by excavating a pit 4×5 m, to the depth of 3 m, and then the previously prepared steel frame was lowered to the bottom of the pit. The excavated material was substituted by the river aggregate whose granulometric composition, and mass content of fractions 0-2, 2-4, 4-8, 8-16 i 16-32 mm in the mixture,

was designed in the Laboratory for geotechnics of the GAF – Niš. The mixture prepared in such a way was installed in layers 30 cm thick, and each layer was compacted by a vibro-plate. Compaction of each layer was determined by testing circular plate. The average values of compaction modulus by layers ranged between 43,3 and 66,7 MPa.

The remaining space between the soil and frame of 0.9m will be used to install the spread footing, hydraulic press and necessary equipment for measuring the applied force. The hydraulic press loaded the foundation with vertical centric force until punching shear of the column through the footing. (Figure 2).

Truss structure of the frame and its dimensions allow for unimpeded formation of sliding surfaces in the soil underneath the footing, in the event sufficiently high failure force is reached. In this way, we have made one step further in respect to the experiments in the literature, because the footings testing is conducted in the completely realistic boundary conditions in terms of the soil. Simultaneously, comparison and verification of earlier tests in laboratories (from the literature) with the *in situ* tests is provided.



Fig. 2 Disposition of the experiment *in situ*

4. MODELING AND ANALYSIS BY FEM

The footing with characteristics given in Figure 3 was chosen for modeling in ANSYS. Compressive strength of concrete footing was $f_{c,\text{cube}}=38.37\text{MPa}$ (150mm cube). Diameter of applied reinforcement was Ø8mm with reinforcement percentage approximately 0.4%. Based on the average values of compaction modulus was adopted modulus of subgrade soil elasticity, which is approximately 40MPa [5].

The adopted material models and mechanical properties of used materials are given in Table 2.

The three additional parameters are necessary to define the non-linear Drucker-Prager model: the cohesion c , angle of internal friction φ and the dilatancy angle ψ . Based on dates at [2] and [4] these parameters are adopted as: $\varphi = 30^\circ$, $\psi = 10^\circ$ and cohesion is calculated from the expression:

$$c = f_c \frac{1 - \sin \varphi}{2 \cos \varphi}$$

where f_c is cylinder compressive strength of concrete.

On the basis of [8] and [9] for tangent modulus of elasticity of bilinear-kinematic model of steel adopted: $E_{steel\ tangent} = 16\% E_{steel}$.

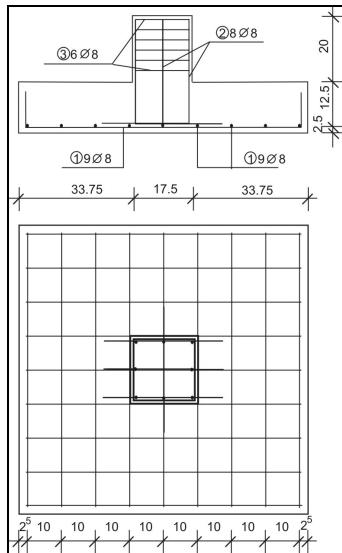


Fig. 3 Characteristics of foundation footing modeled in ANSYS

Table 2 Review of calculation models of materials

Material model description	Basic input data (lin. domain)	Additional input data (nonlin. domain)
Linear elastic model All materials: concrete, steel, soil - Linear elastic	$E_{concrete} = 34.17 \times 10^9 \text{ Pa}, v=0.20$ $E_{steel} = 2.1 \times 10^{11} \text{ Pa}, v=0.30$ $E_{soil} = 40 \times 10^6 \text{ Pa}, v=0.27$	-
Soil: linear elastic Materially nonlinear model Concrete: Drucker-Prager Steel: bilinear-kinematic	$E_{concrete} = 34.17 \times 10^9 \text{ Pa}, v=0.20$ $E_{steel} = 2.1 \times 10^{11} \text{ Pa}, v=0.30$ $E_{soil} = 40 \times 10^6 \text{ Pa}$	$c=19.43 \times 10^9 \text{ Pa}, \phi=30^\circ, \psi=10^\circ$ $E_{steeltangent}=33.6 \times 10^9 \text{ Pa}$

Due to the symmetry of problem in relation to two orthogonal planes was analyzed only fourth basic structure, as shown in Figure 3 and 4. Moving perpendicular to the plane of symmetry is prevented but moving in the other two orthogonal directions is free. For the analysis using FEM created two material models. Concrete and soil are modeled by 3D SOLID65 type elements with three translational degrees of freedom per node, and the reinforcement by line elements LINK8 type. Modeling of reinforcement is done discretely - reinforcing elements are connected to concrete elements in each node, which simulated adhesion and co-operation of concrete and reinforcement. Discretization reinforcing

elements was carried out so that they follow density of the network elements of the concrete. Concrete elements are related to elements of the soil in all the common nodes. For the described models have been performed two FEM analysis - one in a linear domain and one by using of nonlinear model for concrete and steel. 3D model of tested footing with the subsoil is given in Figure 4 and 5.

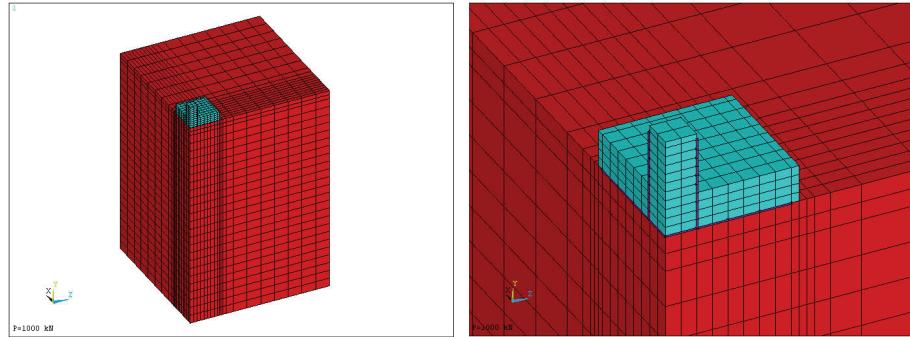


Fig. 4 Whole 3D model - footing and subsoil

Fig. 5 3D model of footing

The distribution of stress and strain was obtained as expected. Figure 6 shows deformed model and distribution of total dilation in z direction, with maximum dilatation of pressure in the upper area of footing, immediately next to the column and maximum tension dilatation in the lower area of footing on the edge of the column below. The display of dilation corresponds to concentrated forces in a column of 900kN. Development of dilatation of concrete (in absolute terms) both for the linear-elastic and elastic-perfect plastic model (Drucker-Prager) during loading of the footing is represented by diagrams in Figures 8 and 9. From the accompanying diagrams it can be seen that the cross-section plastification starts at the load slightly lower than 400kN.

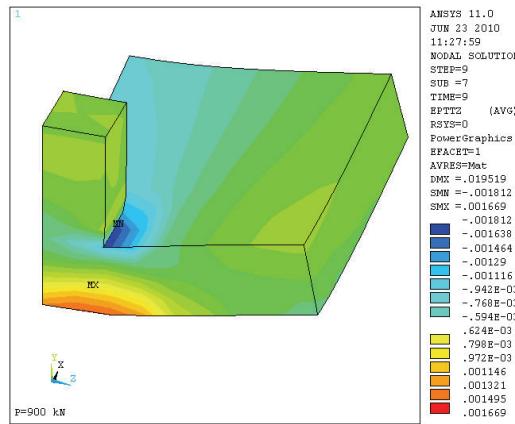


Fig. 6 Total dilatation in z direction in concrete elements at load of 900kN

Total dilatation in z direction in the lower zone of the footing, which matches the force at the beginning of plastification, are represented by Figure 7. It can be seen that the

maximal dilatation are near the projection of the column edge onto the bottom plane of footing. In this section can be expected maximum dilatation in tension reinforcement of footing, because the elements of reinforcement are associated with the finite elements of concrete in each node. Since the dilatation of concrete and reinforcement are equal in the nodes, the values of the dilation at Figure 6 and 7 can be compared with the measured values of reinforcement dilatation during the experiment given in Figure 10. Their very good agreement indicates the quality of the used material model and adopted mechanical characteristics of materials.

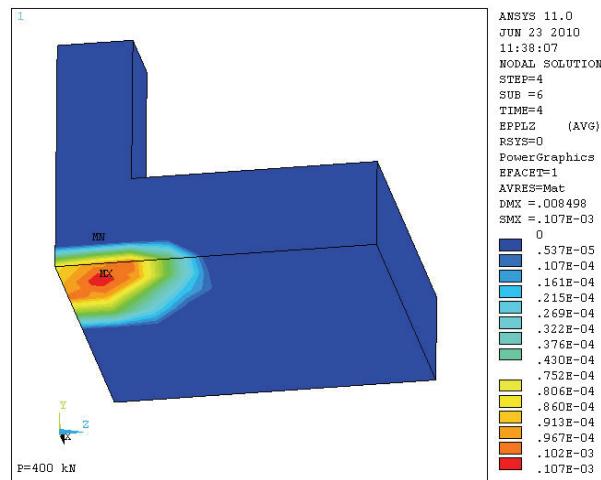


Fig. 7 Total dilatation in z direction in concrete elements at load of 400kN

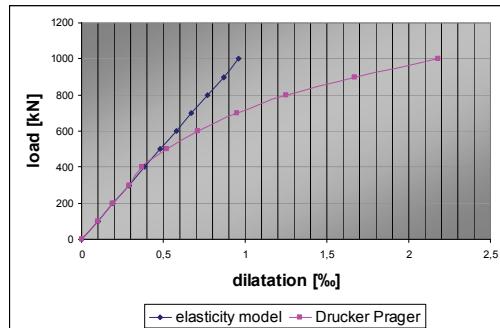


Fig. 8 Total dilatation in z direction in concrete elements in absolute value for both for both models at lower area of footing

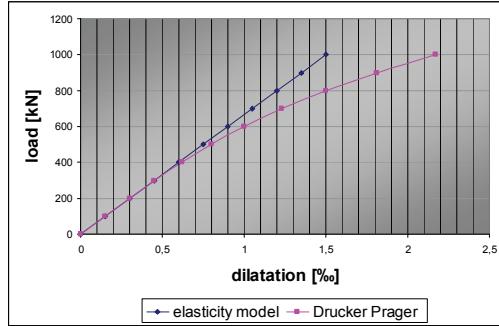


Fig. 9 Total dilatation in z direction in concrete elements in absolute value for both models at upper area of footing

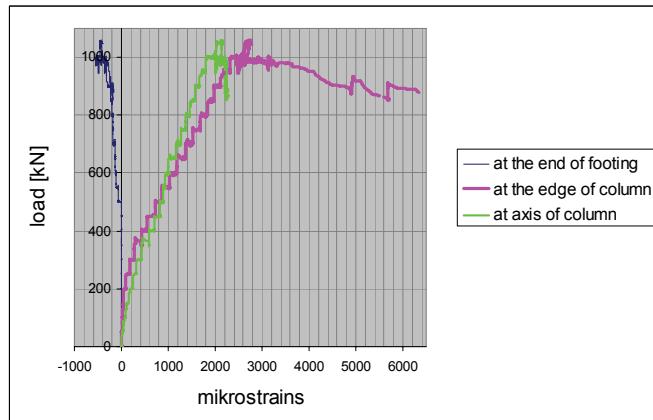


Fig. 10 Dilatation in reinforcement measured in experiment

5. CONCLUSION

Although the establishment of the constitutive relations for defining behavior of materials under external loading is nowadays a very complex process, this problem can be solved primarily owing to development of modern computer technology and software. There are many general-purpose software packages, including the Ansys, which provide almost unlimited possibilities of modeling material nonlinear problems in all areas of technology. The example of a spread footing processed by Ansys is only a small part of the possibilities that this software package provides. Here the appearance of a load level corresponding to plastic deformations, the deformations development with increasing load, as well as the distribution of dilatation in the footing, have been observed. Comparison of calculated and measured values of reinforcement dilatation shows their very good agreement.

Acknowledgement. *The paper is result of the investigation in the project TR 16021 financed by the Ministry of Science and Technological Research of Republic of Serbia*

REFERENCES

1. Ansys Inc. Theory Reference – Ansys Manual, <http://www.ansys.com>.
2. Lopez Cela J.J.: Material Identification Procedure for Elastoplastic Drucker–Prager Model, Journal of engineering mechanics, may 2002, pp. 586-591.
3. Maneetes H., Memari A. M.: Finite Element Modeling of Reinforced Concrete Cladding Panels, Electronic Journal of Structural Engineerin, 9 (2009) 62-72.
4. Marinković S.: Granična nosivost pri probijanju montažnih prethodno napregnutih ploča u oblasti ivičnih stubova, doktorska disertacija, Gradevinski fakultet Beograd, 2000.
5. Prolović V., Bonić Z.: The Calculation of Foundation Girders on Equivalent Elastic Semi-Space, Facta Universitatis, Series Architecture and civil engineering, Vol. 2 N° 1, pp. 61-66, University of Niš, 1999
6. Prolović V., Bonić Z.: Smičuća nosivost na probijanje plitkih temelja u domaćoj i stranoj regulativi, Zbornik radova Gradevinsko-arkitektonskog fakulteta u Nišu no.23, 2008, pp. 93-103
7. Vacev T., Prolović V., Bonić Z., Kajganović T.: Konstruisanje i proračun čeličnog rama za ispitivanje plitkih temelja na deformabilnoj podlozi, Zbornik radova Gradevinsko-arkitektonskog fakulteta u Nišu no.24, 2008, pp. 79-88, 2010
8. Vacev T., Kisin S., Ranković S.: Experimental analysis of an original type of steel space truss node joint, Facta Universitatis, Series Architecture and civil engineering, Vol. 7 N° 1, pp. 43-55, University of Niš, 2009
9. Vacev T.: Optimalno rešenje čvora čelične prostorne rešetke primenom nelinearne analize, doktorska disertacija, Fakultet tehničkih nauka u Novom Sadu, 2009.

MATEMATIČKO MODELIRANJE MATERIJALNO NELINEARNIH PROBLEMA U ANALIZI KONSTRUKCIJA (II DEO – PRIMENA U SAVREMENOM SOFTVERU)

**Zoran Bonić, Todor Vacev, Verka Prolović,
Marina Mijalković, Petar Dančević**

U radu je data primena nelinearnih modela materijala u softverskom paketu Ansys. Razvoj teorije modela je predstavljen u radu pod nazivom matematičko modeliranje nelinearnih problema materijala u analizi konstrukcija (deo I – teorijske osnove), a ovde je opisana inkrementalno-iterativna procedura za rešavanje problema materijalne nelinearnosti koja se koristi u ovom paketu i dat primer modeliranja temelja samca korišćenjem Bilinearno-kinematičkog i modela Drucker-Pragera. Uradena je komparativna analiza rezultata dobijenih primenom ovih modela sa eksperimentalnim istraživanjima autora ovog rada. Uočen je nivo opterećenja koji odgovara pojavi plastičnih deformacija, njihov razvoj sa porastom opterećenja kao i raspodela dilatacija u temelju. Poređenje tako sračunatih i izmerenih vrednosti dilatacija armature pokazuje njihovo veoma dobro slaganje.

Ključne reči: *modeli materijala, nelinearnost, Bilinearno kinematički, Drucker Prager, eksperiment*