USE OF FINITE ELEMENT METHOD FOR SIMULATION OF RC BEAM NONLINEAR BEHAVIOR

Andrija Rašeta[1]
Igor Džolev[2]
Anka Starčev-Ćurčin[3]
Đorđe Lađinović[4]
Danijel Kukaras[5]

UDK: 624.072.2:517.9
DOI:10.14415/konferencijaGFS2017.024

Summary: *In this paper, nonlinear behavior of reinforced concrete beam, subjected to monotonically increasing load and designed according to EN 1992-1-1, was analyzed. For discretization of the concrete member part, 3-dimensional finite elements, SOLID65, in the program Ansys, 2-dimensional finite elements, CPS4, in the program Abaqus and 2-dimensional finite elements, 2D Solid, in the program ADINA were applied. The reinforcement was modeled using the basic rod and beam finite elements. Numerical analyzes were done on the spatial model in the program Ansys and simplified two-dimensional models with the assumption of a plane stress condition in Abaqus and ADINA programs. The results of the analysis of the stress-strain state and the vertical displacement of the beam mid-span are presented.*

Keywords: *reinforced concrete, nonlinear analysis, finite element method*

1. INTRODUCTION

A large number of different numerical models, implemented in various programs based on Finite Element Method, are developed for reinforced concrete as a complex material. Numerical models need to adequately cover the behavior of concrete and reinforcement, as well as their interaction. Material nonlinearity can be include in the models with the so-called concentrated and distributed plasticity. Models with concentrated plasticity include the formation of plastic hinges in certain sections and the behavior of a plastic joint is usually defined via the force-displacement or moment-rotation relation. In the models with distributed plasticity, a fiber cross-section models that simulate distributed plasticity on the surface of the cross section and along the length of the element in the respective integration points, are used. Distributed plasticity models can be implemented with the displacement-based finite elements formulations (Hellesland, Scordelis 1981; Marí, Scordelis 1984), or with the force-based formulations (Spacone et al. 1996; Neuenhofer,

---

[1] PhD, Faculty of Technical Sciences, University of Novi Sad, Serbia, e – mail: araseta@uns.ac.rs
[2] MSc, Faculty of Technical Sciences, University of Novi Sad, Serbia, e – mail: igordzolev@uns.ac.rs
[3] MSc, Faculty of Technical Sciences, University of Novi Sad, Serbia, e – mail: astarcev@uns.ac.rs
[4] PhD, Faculty of Technical Sciences, University of Novi Sad, Serbia, e – mail: ladjin@uns.ac.rs
[5] PhD, Civil Engineering Subotica, University of Novi Sad, Serbia, e – mail: danijel.kukaras@gmail.com

Filippou 1997). The above-described simplified models of reinforced concrete behavior have an important application in the analysis of building structures subjected to seismic load and have been implemented in many programs, such as SAP2000 and SeismoStruct. In addition to the simplified models of reinforced concrete behavior, advanced models, so-called Continuum mechanics-based models, can be used. In these models, the concrete is discretized with 3-dimensional solid finite elements, and reinforcement using bar or beam finite elements. The main priority in the application of such models is that the triaxial stress state in almost all types of reinforced concrete structures can be take into account, as well as all models of fracture (brittle shear fracture, concrete grinding, cracking, etc.), including the interaction between all forms of degradation in the element (eg. damage to the concrete and the reinforcement dut to buckling, the interaction between the axial and bending stresses in the element, etc.) that cannot be "easily" included with the application of simplified models. In this paper three different models of concrete behavior are applied and implemented in the Ansys, Abaqus and ADINA programs. The models are concisely presented in the following chapter and described in detail in [1], [5] and [7]. Also, all analysis are made with the assumption of no bond slip between reinforcement and concrete.

2. APPLIED MODELS FOR CONCRETE AND REINFORCEMENT

The first model of concrete and reinforcement behavior applied in this paper is implemented in the Ansys program [1]. For the modeling of concrete member part, 3-dimensional finite element SOLID65 [1] is applied, and for the modeling of the reinforcement finite element BEAM188 [1] is used. The geometric characteristics of the applied finite element are shown in Fig. 1.

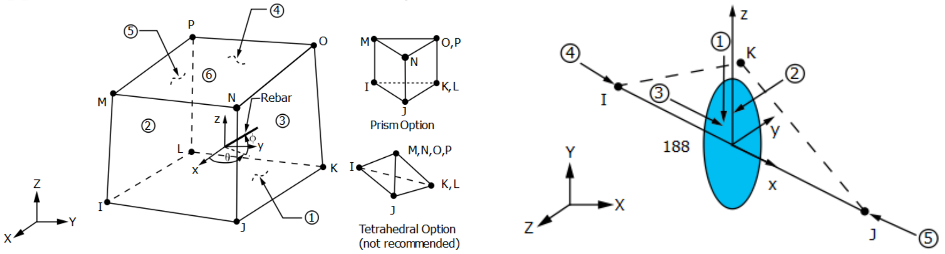

Fig. 1. 3-D element SOLID65 (left) and BEAM188 element (right) [1]

The concrete material is capable of plasticity, creep, cracking and crushing [2], [3] and [4]. Cracking is permitted in three orthogonal directions at each integration point and it is modeled through an adjustment of material properties which effectively treats the cracking as a “smeared band” of cracks. In addition to cracking and crushing, the concrete may also undergo plasticity, with the Drucker-Prager failure surface being most commonly used. In this case, the plasticity is done before the cracking and crushing checks. The presence of a crack at an integration point is represented through modification of the stress-strain relations by introducing a plane of weakness in a direction normal to the crack face. Fig. 2 shows the stress-strain relation of concrete in uniaxial tension. If the material at an integration point fails in uniaxial, biaxial, or triaxial compression, the material is assumed to crush at that point. The concrete material model predicts the failure of brittle materials and both cracking and crushing failure modes are accounted for. The criterion for failure of concrete due to a multiaxial stress state is given in [2]. Fig. 3 shows Failure Surface in Principal Stress Space for Biaxial Stress (left) and 3-D Failure Surface in Principal Stress Space (right).

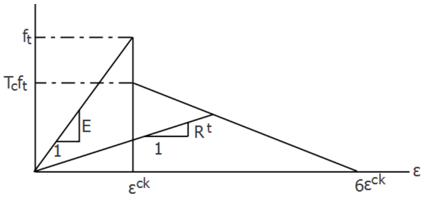

Fig.2. Uniaxial model of the concrete behavior in tension [1]

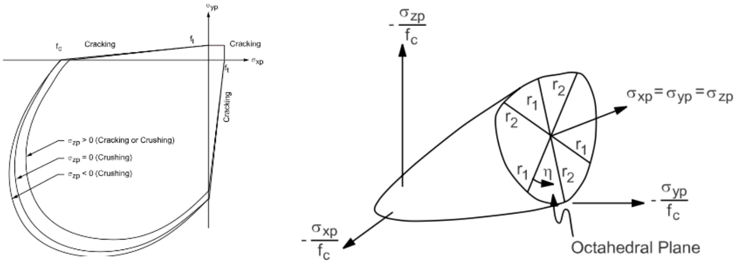

Fig.3. Biaxial (left) and triaxial (right) failure envelope [1]

The second model of concrete behavior used in this paper is implemented in the program Abaqus with the name Concrete Damaged Plasticity (CDP) [5]. This model is a continuum, plasticity-based, damage model for concrete, that provides a general capability for modeling concrete and other quasi-brittle materials in all types of structures (beams, trusses, shells and solids) and uses concepts of isotropic damaged elasticity in combination with isotropic tensile and compressive plasticity to represent the inelastic behavior of concrete. It is designed for applications in which concrete is subjected to monotonic, cyclic, and/or dynamic loading under low confining pressures. This model consists of the combination of nonassociated multi-hardening plasticity and scalar (isotropic) damaged elasticity to describe the irreversible damage that occurs during the fracturing process. The plastic-damage model in Abaqus is based on the models proposed by Lubliner et al. (1989) and by Lee and Fenves (1998) [6]. The main failure mechanisms are cracking in tension and crushing in compression. This model assumes non-associated potential plastic flow and the flow potential used for this model is the Drucker-Prager hyperbolic function. The yield function, represents a surface in effective stress space, which determines the states of failure or damage. The degraded response of concrete is characterized by two independent uniaxial damage variables, d_c and d_t (Fig. 4) which can take values in the range from zero (undamaged material) to one (fully damaged material). Concrete behavior in uniaxial tension and pressure is shown in Fig. 4, and yield surfaces, in the deviatoric plane (left) and yield surface in plane stress (right), are shown in Fig. 5. The parameters in this model are: K_c – the ratio of the second stress invariant on the tensile meridian to that on the compressive meridian or in another word the ratio between the magnitude of the deviatory stress in uniaxial tension to uniaxial compression ($0.5 < K_c \leq 1.0$), ψ – dilation in a frictional material is the phenomenon of the inelastic volume change due to plastic deviation (~ 13° - 56°), f_{bo}/f_{co} – ratio of the biaxial and uniaxial compressive strengths (1.16), ϵ – eccentricity of the plastic potential surface (0.1) and μ – viscosity parameter representing the relaxation time of the viscoplastic system (0). For the modelling of concrete member part 2-dimensional finite element CPS4 [5] is applied (assumption of plane stress), and for reinforcement modelling finite element T2D2 [5] is used.

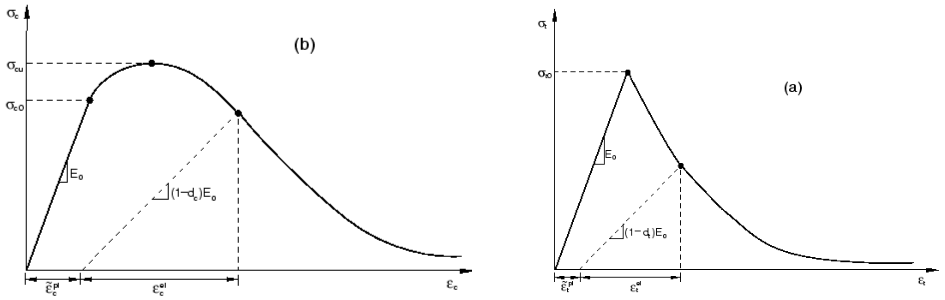

Fig.4. Response of concrete to uniaxial loading in compression (left) and tension (right) [5]

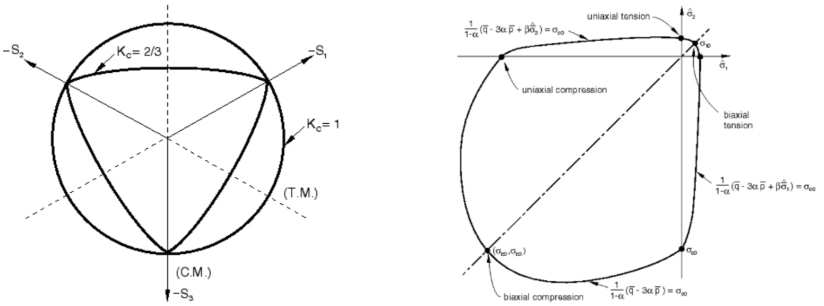

Fig.5. Yield surfaces in the deviator plane (left) and yield surface in plane stress (right) [5]

The third model of concrete behavior that was used in this paper is implemented in the ADINA program with the name Concrete [7]. The basic material characteristics of this model are: tensile cracking failure at a maximum, relatively small principal tensile stress, compression crushing failure at high compression and strain softening from compression crushing failure to an ultimate strain, at which the material totally fails and the tensile cracking and compression crushing failures are governed by tensile failure and compression crushing failure envelopes. The general multiaxial stress-strain relations are derived from a uniaxial stress-strain relation (Fig. 6) [7]. The failure envelopes are employed to establish the uniaxial stress-strain law accounting for multiaxial stress conditions, and to identify whether tensile or crushing failure of the material has occurred. The triaxial compressive failure envelope, the biaxial compressive failure envelope, curve 1, and three-dimesional tensile failure envelope are shown in Fig. 7 [7] and [8]. To identify whether the material has failed, the principal stresses are used to locate the current stress state. The tensile strength of the material in a principal direction does not depend on tensile stresses in the other principal stress directions, but depends on compressive stresses in the other directions. For the modelling of concrete member part finite element 2-D Solid with nine nodes is applied [7] and [9] (assumption of plane stress), and for the modelling of reinforcement finite element Truss, [7] and [9], is used. In all analysis in this paper, bilinear model for stress-strain relation of the reinforcement is applied.

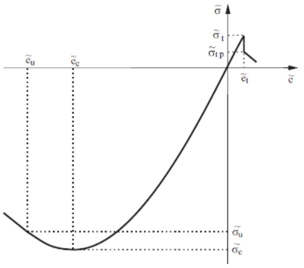

Fig.6. Uniaxial stress-strain relation [7]

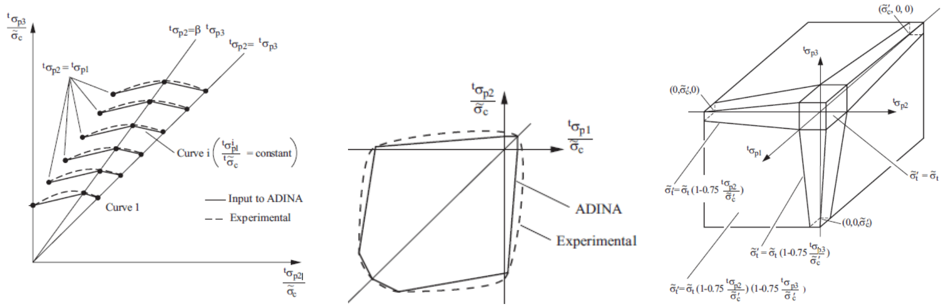

Fig.7. Triaxial compressive (left), biaxial compressive (middle) and three-dimensional tensile (right) failure envelope [7]

3. NUMERICAL ANALYSIS

In this paper the simple supported reinforced concrete beam exposed to self weight, and two concentrated forces symmetrically arranged in regard to the middle span, is analyzed. It is assumed that the load is applied slowly enough that it can be considered as static.

The analyzed reinforced concrete beam is designed according to EN 1992-1-1 [10]. The characteristics of the concrete member part are: C25/30, $f_{ck} = 25$ MPa; $f_{cm} = 33$ MPa; $f_{ctm} = 2.6$ MPa; $\varepsilon_{c1} = 0.0021$; $\varepsilon_{cu1} = 0.0035$, and the characteristics of the reinforcement are: B500B, $f_{yk} = 500$ MPa, $E = 200$ GPa. The geometrical characteristics of the analyzed beam are shown in Fig. 8.

Stress-strain relations for concrete and reinforcement are shown in Fig. 9 in the form of Engineering Stress – Engineering Strain. The model parameters of the concrete stress-strain relation in tension in Ansys was determined according to the recommendations given in [1] (Fig. 10 – left) and for Abaqus was determined according to [5] and [12] with modifications shown in Fig. 10 (middle). In ADINA program, behavior of concrete in tension is defined by fracture energy parameters [7] with adopted value G_f = 0.0675 N/mm according to the recommendations given in [11] for maximum aggregate size d_{max} = 16 mm.

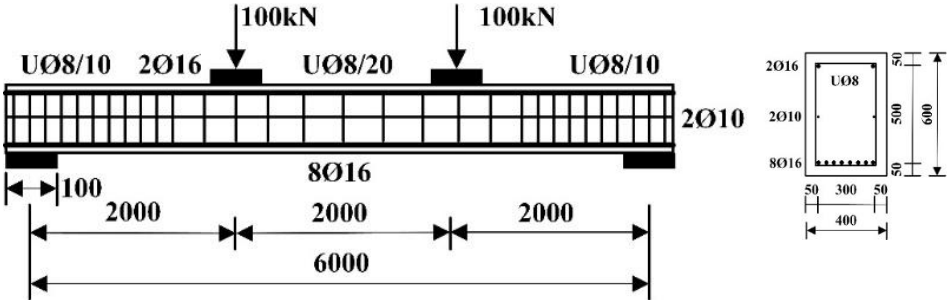

Fig.8. RC beam

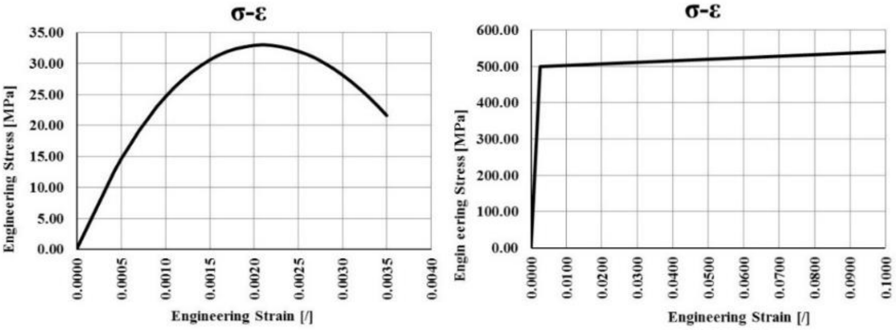

Fig.9. Stress-strain relation in concrete for compression (left) and in reinforcement (right) in the form of Engineering Stress – Engineering Strain [10]

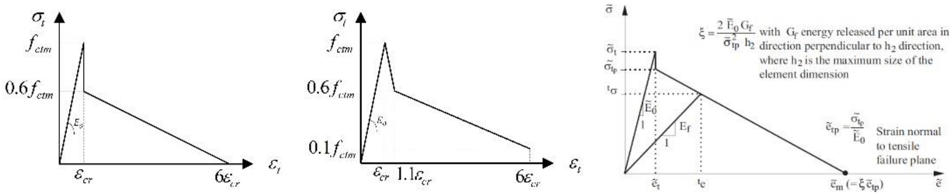

Fig.10. Stress-strain relation in concrete for tension (Ansys – left, Abaqus – middle and ADINA – right [7])

Numerical analysis on the 3-dimensional model was done in Ansys program using member symmetry (Fig. 11 – left). Finite element mesh for concrete part of the member (size of the finite element is 25x25x25 mm) and for reinforcement (size of the finite element is 25 mm) is shown in Fig. 11.

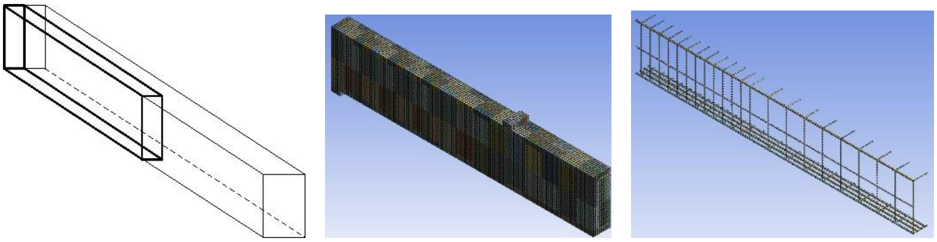

Fig.11. Member symmetry (left), finite element mesh for concrete member part (middle) and finite element mesh for reinforcement (right) – Ansys

Numerical analyses in Abaqus and ADINA were done on a model with the assumption of plane stress state using member symmetry (Fig. 12). Finite element mesh for concrete part of the member (size of the finite element is 25x25 mm) and for reinforcement (size of the finite element is 25 mm) is shown in Fig. 13. Characteristics of the applied CDP model in Abaqus program are determined according to the recommendations given in [13] and [14]: $\psi = 13°$, $\epsilon = 0.1$, $f_{bo}/f_{co} = 1.16$, $K_c = 2/3$ and $\mu = 0$.

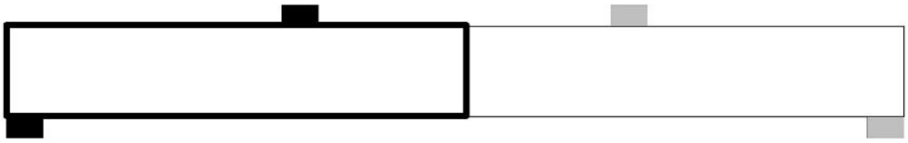

Fig.12. Member symmetry used in the analysis with Abaqus and ADINA programs

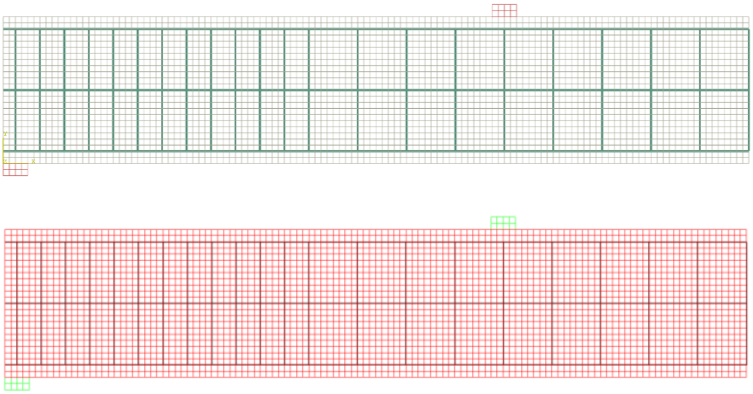

Fig.13. Finite element mesh (up – Abaqus and down – ADINA)

The vertical displacements graph of the middle bottom beam edge is shown in Fig 14., and damage arranges caused by tension in the concrete member part (cracks) for all applied models are shown in Fig. 15. The maximum stress and strain values of the reinforcement and the maximum displacement values of the middle bottom member edge are shown in Tab. 1.

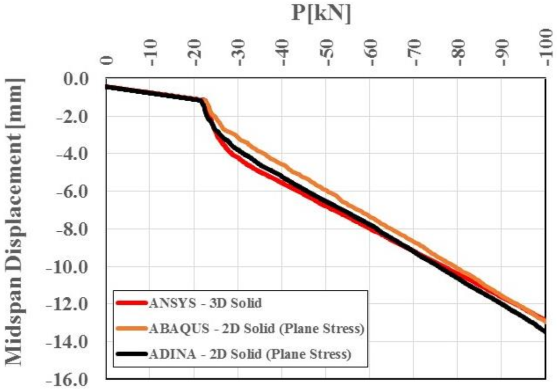

Fig.14. Vertical displacements of the middle bottom beam edge

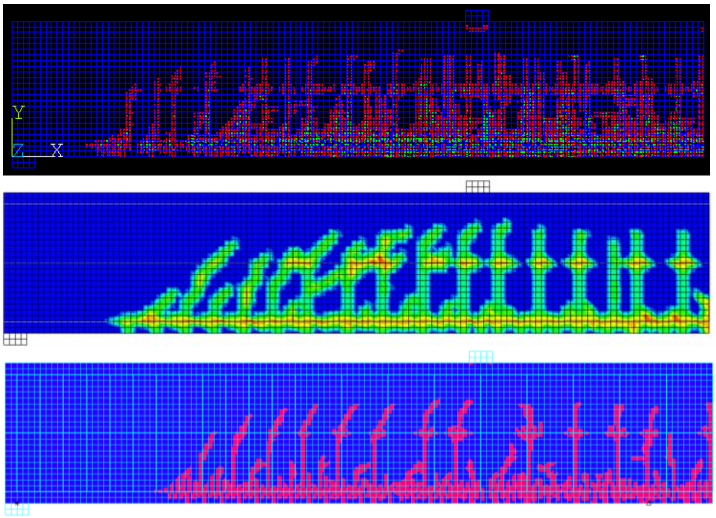

Fig.15. Damage arranges caused by tension in the concrete member part (cracks) – Ansys (up), Abaqus (middle) and ADINA (bottom)

Table 1. The maximum stress and strain values of the reinforcement and the maximum displacement values of the middle bottom RC member edge

	Max. Stress [MPa]	Max. Strain [‰]	Ugib [mm]
Ansys (3D)	276	1.380	12.9
Abaqus (2D – Plane Stress)	254	1.270	13.0
ADINA (2D – Plane Stress)	264	1.320	12.7
“Hand” design (EN 1992-1-1)	285	1.425	12.8

Based on the analysis results given in Fig 14. and in Tab. 1 it can be concluded that the differences in stresses and strains of the reinforcement between the 3-dimensional models and models with the assumption of plane stress are not more than about 8%, in absolute terms. Differences between the maximum vertical displacement values of the middle bottom beam edge are not greater than about 2%, by absolute value. Also, differences in the results for the stress and strain of the reinforcement between the "hand" design and the applied models do not exceed about 11%, by absolute value. Differences between the maximum vertical displacements of the middle bottom beam edge do not exceed 2%, in absolute terms.

4. CONCLUSION

In this paper, three different nonlinear material models for concrete are applied and implemented in Ansys, Abaqus and ADINA programs. The models are used for nonlinear analysis of reinforced concrete simply supported beam subjected to monotonous increasing load. Also, two types of analysis were applied. The first, used for the three-dimensional model in Ansys program and the other with the adopted assumption of plane stress in Abaqus and ADINA programs.

Based on the results of the analysis it can be concluded that the differences in stresses and strains between the 3-dimensional models and models with the assumption of plane stress are not higher than about 8%. Also, differences in the results for the stress and strain of the reinforcement between the "hand" design and applied models in programs Ansys, Abaqus and ADINA do not exceed about 11%. Analysis of maximum vertical displacements of the middle bottom RC beam edge shows that the differences between the calculated values do not exceed about 2%.

Based on the above, it can be concluded that in the cases, where it is possible, the simplified models can be used compared to the three-dimensional, because the models with 3D solid finite elements are much more computationally expensive, compared to the models with 2D finite elements that gives satisfactory results with engineering accuracy.

ACKNOWLEDGEMENTS

The research has been conducted within the scientific research project TR 36043, funded by the Ministry of Science of Serbia.

REFERENCES

[1] ANSYS® Academic Teaching Mechanical: ANSYS Help Documentation, Release 17.0, Canonsburg (PA), ANSYS, Inc., **2015**.

[2] K. J. Willam and E. D. Warnke. "Constitutive Model for the Triaxial Behavior of Concrete". Proceedings, International Association for Bridge and Structural Engineering. Vol. 19. ISMES. Bergamo, Italy. p. 174. **1975**.

[3] E. L. Wilson, R. L. Taylor, W. P., Doherty, and J. Ghaboussi. "Incompatible Displacement Models". Numerical and Computer Methods in Structural Mechanics. edited by S. J. Fenves, et al. Academic Press, Inc.. N. Y. and London. 43-57. **1973**.

[4] R. L. Taylor, P. J. Beresford, and E. L. Wilson. "A Non-Conforming Element for Stress Analysis". International Journal for Numerical Methods in Engineering. Vol. 10. 1211-1219. **1976**.
[5] ABAQUS 2016 Theory Guide, © Dassault Systèmes, **2015**.
[6] Lee, J., and G. L. Fenves: Plastic-damage model for cyclic loading of concrete structures, J. Eng. Mechanics 124(8), pp. 892-900, **1998**.
[7] ADINA Theory and Modeling Guide, ADINA R & D, Inc. 71 Elton Avenue Watertown, MA 02472 USA, September **2016**.
[8] Kupfer, H. B., and K. H. Gerstle, "Behaviour of concrete under biaxial stresses," Journal of Engineering Mechanics Division 99 (EM4), pp. 853-866, **1973**.
[9] K.J. Bathe: Finite Element Procedures, Prentice Hall, Englewood Cliffs, NJ, **1996**.
[10] EN 1992-1-1: Design of Concrete Structures, General Rules and Rules for Buildings, European Committee for Standardization, **2004**.
[11] CEB-FIP Model Code 1990, Comite Euro International du Beton, **1993**.
[12] Nayal, R., & Rasheed, H.A. (**2006**). Tension Stiffening Model for Concrete Beams Reinforced with Steel and FRP Bars. Journal of Materials in Civil Engineering, 18(6), 831-841.
[13] Francisco Lopez-Almansa, Bashar Alfarah, Sergio Oller: Numerical Simulation of RC Frame Testing With Damaged Plasticity Model Comparison With Simplified Models, Secnd Europen Conference on Erthquake Engineering And Seismology, Isntanbul, Aug. 25-29, **2014**.
[14] Sergio Oller: Nonlinear Dynamics of Structures, Springer, ISBN: 978-3-319-05194-9 (e-book), **2014**.

PRIMENA METODE KONAČNIH ELEMENATA ZA SIMULACIJU NELINEARNOG PONAŠANJA AB GREDE

Rezime: *U ovom radu analizirano je nelinearno ponašanje armiranobetonske grede izložene monotono rastućem opterećenju koja je dimenzionisana prema EN 1992-1-1. Za diskretizaciju betonskog dela nosača primenjeni su 3-dimenzionalni konačni elementi SOLID65 u programu Ansys, 2-dimenzionalni konačni elementi CPS4 u programu Abaqus i 2-dimenzionalni konačni elementi 2D-Solid u programu ADINA. Armatura je modelirana primenom osnovnih štapnih i grednih konačnih elemenata. Numeričke analize urađene su na prostornom modelu u programu Ansys i pojednostavljenim dvodimenzionalnim modelima sa pretpostavkom ravnog stanja napona u programima Abaqus i ADINA. Prikazani su rezultati analiza naponsko-deformacijskog stanja i vertikalnog pomeranja sredine raspona grede.*

Ključne reči: *armirani beton, nelinearna analiza, metod konačnih elemenata*