Mathematical and Computational Applications, Vol. 16, No. 4, pp. 947-958, 2011.

NONLINEAR FINITE ELEMENT ANALYSIS OF AN R/C FRAME UNDER LATERAL LOADING

Yunus Dere
Department of Civil Engineering
Selcuk University
42075 Konya, Turkey
ydere@selcuk.edu.tr

Fatma Tuba Dede
Sari Engineering
Kemerli Is Merkezi A Blk K:4/408
Selcuklu, Konya, Turkey
ftdede81@hotmail.com

Abstract- Recent developments in computer technology have made possible the use of finite element methods for 3D modeling and analysis of reinforced concrete structures. In this study, the failure behavior and crack formation of an R/C frame under monotonic and reversed-cyclic lateral loading are studied by 3D nonlinear finite element analysis using ANSYS software. Modeling the nonlinear behavior of concrete material as well as the reinforcing steel embedded within concrete is a difficult task. Different methodologies and modeling options are considered in the computer model. The application of reversed cyclic displacement loading and the execution of nonlinear analyses are explained in detail. Load-displacement relationships and concrete crack profiles are obtained in order to compare with the experimental data and observed crack profile. The analysis results compared well with the experimental data based on the comparison of load-displacement graphs. The failure mode of the frame is identified by the crack profiles displayed on the structure.

Keywords- Failure analysis, Cracks, Finite element analysis, Reinforced concrete

1. INTRODUCTION

In order to properly design Reinforced Concrete (R/C) structures to withstand lateral loads resulting from earthquakes as well as high winds, the failure behavior is usually investigated through laboratory tests. Due to the difficulties faced during the fabrication and testing of real size models of R/C buildings and frames within a laboratory, Finite Element (FE) analysis of the computer models of R/C structures is quite popular among researchers. Moreover, computer simulations of planned experiments prior to laboratory fabrication and testing processes provide significant advantages such as reduced amount of construction materials, labor and time. However, the nonlinear stress-strain relationship of concrete, time dependent deformations such as creep and shrinkage, aggregate interlock, tension cracks, compression failure and the adhesion between concrete and reinforcing steel cause difficulties in the modeling of R/C members. There are many commercially available FE software packages capable of modeling nonlinear behavior of materials such as ABAQUS, ANSYS, DIANA, MSC NASTRAN and ADINA. This work is focused on FE modeling and analysis of R/C frames with ANSYS [1] since it is a sophisticated software with a 3D eight-node isoparametric FE element which can be used to model concrete material and its cracking behavior. ANSYS modeling of reinforced concrete structures has been studied by many researchers [2-12]

In this study, the FE model of an R/C frame fabricated and experimentally tested by Akin [13] at Selcuk University, Civil Engineering Structures Laboratory is constructed and analyzed under monotonic and reversed-cyclic lateral loading. The process of constructing the model, modeling and analysis parameters and options required to be specified and the concerns related to the convergence of the analyses are described. The failure behavior is investigated through load and displacement relationships obtained from analyses and which are compared later with the experimental data. Moreover, the crack profiles obtained at various stages of the reversed-cyclic loading are significant in the identification of the failure modes of the R/C frames.

2. MODELING REINFORCED CONCRETE MEMBERS

The *Solid65* element type (Figure 1) available in ANSYS element library is a solid element with 8 nodes and has three translation degrees of freedom at each node. It can be used to model concrete with or without rebars and is capable of modeling cracking in tension, crushing in compression, plastic deformation and creep behavior.

Rebars are smeared within the element and can be defined in three different axes. Full adherence between concrete and rebar and is assumed. *Solid65* can crack in three orthogonal axes at each Gauss integration point. As a result, the maximum number of cracks in an element is 8x3=24. These cracks are labeled as “first”, “second” and “third” according to their order of occurrence. Analysis crack profiles can be visualized based on their labels. Crack widths are not supplied by the element and therefore existence of a crack does not mean that it is visible to the eye. Crack profiles with “third” cracks are usually more consistent with the experimental ones than the ones with “first” and “second” cracks. An integration point with a “third” crack is cracked at all three axes; therefore the crack width is probably larger and more visible.

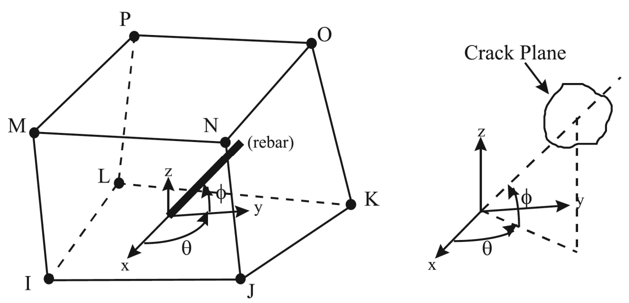

Figure 1. *Solid65* element type as defined in ANSYS

2.1. Concrete material model

The concrete material model in ANSYS uses a failure model developed by Willam and Warnke [14] for multi-axial stress state. *Solid65* element decides the cracking and crushing of concrete through this material model. A material model may be composed of two or more material definitions. Concrete material should have at least *Elastic* and *Concrete* material definitions. In *Elastic* definition, the modulus of elasticity and Poisson's ratio are necessary. The modulus of elasticity of concrete can be determined by either experiments or existing formulations. For *Concrete* definition, axial tension strength of concrete and shear transfer coefficients between crack surfaces for open and closed cracks are required. If the shear transfer from one crack surface to the other does not exist then the shear transfer coefficient is 0.0, if it fully exists then the coefficient is 1.0. In the literature, there are different suggestions for this coefficient by researchers.

ANSYS does not allow for the definition of an additional material model for the tension behavior of concrete. However, if requested/required, an additional stress-strain relationship for compressive behavior can be defined through a hardening model such as *Multilinear Isotropic Hardening*. If this is the case, the modulus of elasticity must be the same as the slope of the initial tangent of the defined stress-strain curve.

2.2. Modeling the reinforcement

The rebars can be either smeared within the *Solid65* elements or defined discretely by spar elements (such as *Link8*) attached to the nodes of *Solid65* elements. If the smeared rebar model is used, then the reinforcement should be defined using the real constants of *Solid65* element.

Confinement reinforcement, in addition to the stated methods, can also be included in the concrete material model for the confined concrete region. There is no need to model rebars using smeared or discrete approaches, if confined concrete stress-strain relationship given by one of the existing mathematical models is used in the concrete model. During the modeling of confinement reinforcement, this method requires usually less effort than discrete or smeared modeling.

3. FE MODELING AND FAILURE ANALYSIS OF AN R/C FRAME

The 3D FE model of an R/C frame designed and tested by Akin [13] at Selcuk University, Civil Engineering Structures Laboratory is created using ANSYS. The R/C frame specimen consisting of three stories and three bays is subjected to reversed-cyclic loading applied at the top story level. The FE model is initially analyzed under monotonic lateral loading. It is later subjected to reversed cyclic loading analysis.

3.1. The properties of the frame specimen

The geometry, section and reinforcement details of the R/C frame are shown in Figure 2. Column and beam dimensions are 120x120mm and 120x150mm, respectively. The frame is arranged symmetrically and the middle bay is constructed narrower than the side bays. At the beam-column connections of the frame, confinement reinforcement was not placed on purpose. The columns are reinforced with 4ϕ7 straight bars, and beams are reinforced with 2ϕ6 straight bars, 2ϕ6 top bars to hold the stirrups and 1ϕ6 bent bar. ϕ4/6 stirrups are used as the confinement reinforcement in both columns and beams. The characteristic compression strength of concrete is 180kg/cm^2 and the yield strength of steel is 5200 kg/cm^2.

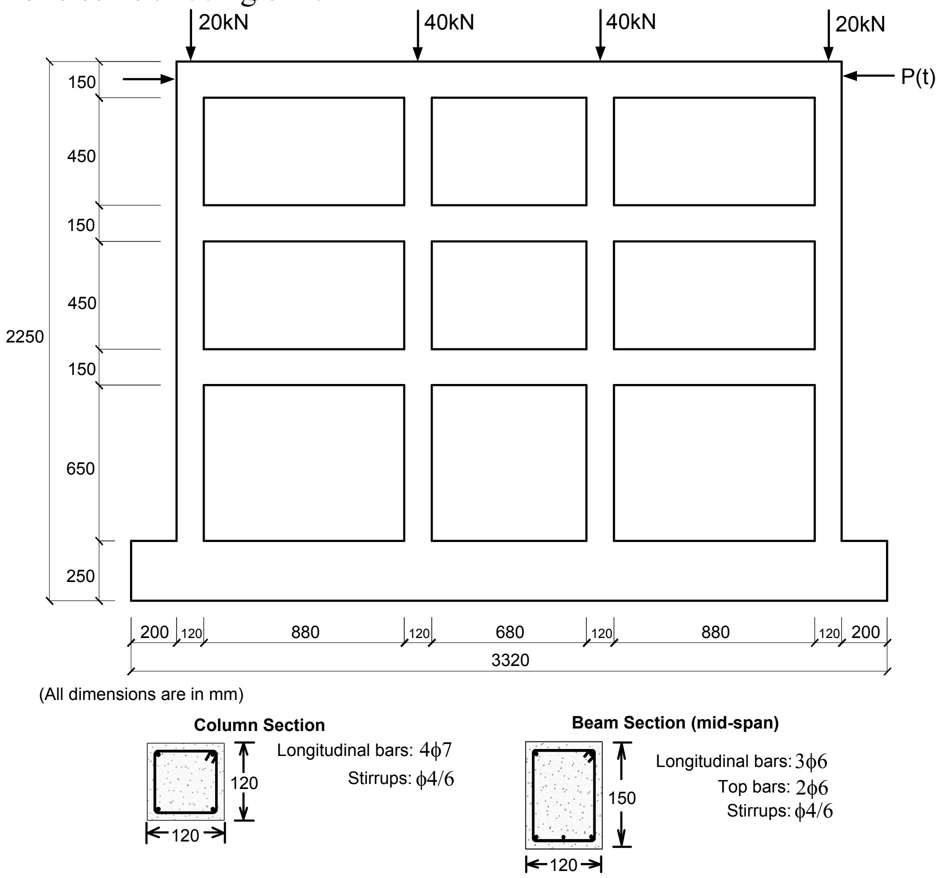

Figure 2. The geometry, cross-sections and rebar details of the R/C frame

3.2. FE Model of the Frame

In ANSYS, the frame model created with *Volume* objects in consistence with the frame's geometry is meshed with *Solid65* type finite elements. Mesh discretization is performed in a way that the longest edge of an element should not exceed 30 mm. As a result, the frame model is made up of 8080 elements and 12500 nodes. The FE mesh, support restraints at the bottom of the columns and the load application points at the top story level are shown in Figure 3. Since *Solid65* element type does not have rotational degrees of freedom, bending deformations along the columns under horizontal loading may be computed with some errors depending on the element edge dimensions. However, for the existing model, the chosen element edge length was verified to be sufficient by comparing the experimental and analytical results.

In the FE model of the frame, the straight rebars in the beams and columns are represented using smeared rebar option for the *Solid65* elements located closest to the physical rebar positions. Therefore discrete elements are not required to model the reinforcement. The properties of the smeared rebars are given using *element real constants* (Table 1). It is necessary to define a separate real constant for every reinforcement ratio and orientation. In Figure 4, the real constant numbers at a typical beam column connection is shown.

The stress-strain relationship of the steel reinforcement is defined using *Bilinear Isotropic Hardening* option (Figure 5). The yield strength and modulus of elasticity of steel are assumed as 5,200 kg/cm^2 and 2,000,000 kg/cm^2, respectively. Therefore the yielding strain is computed as 0.0026. In order to help overall convergence of the FE analyses, the slope of the second line segment is assumed to be 1,000 kg/cm^2.

Table 1. The element *real constants* defined for *Solid65* element type.

Real Constant	Material Number	Ratio	θ^o	ϕ^o	Explanation
1	-	-	-	-	No reinforcement
2	2 (Steel)	0.0428	90	0	Column reinforcement
3	2 (Steel)	0.0248	0	0	Beam reinforcement
4	2 (Steel)	0.0428 0.0248	90 0	0	Beam-column reinforcement

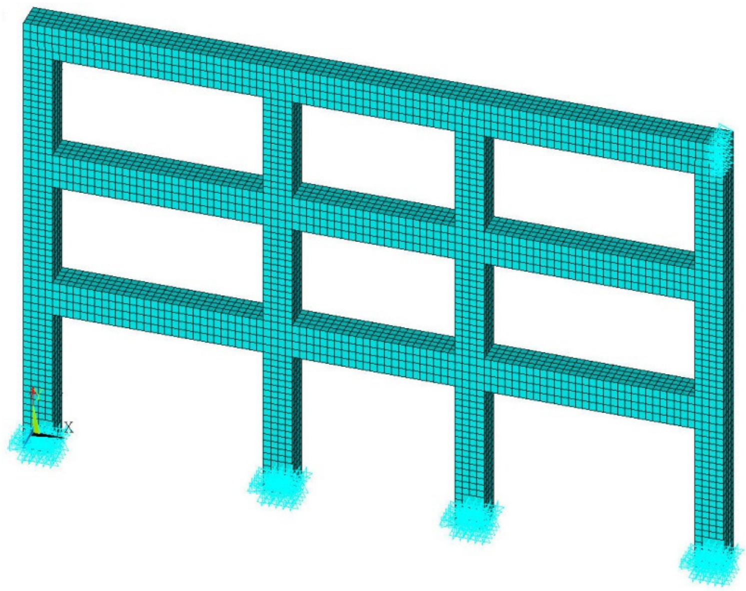

Figure 3. The FE mesh of the R/C frame model

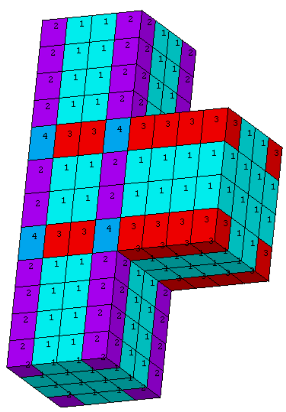

Figure 4. The real constant numbers at the beam-column connection

The stress-strain relationship of concrete is modeled through the *Multilinear Isotropic Hardening* option (Figure 6). In order to obtain the stress-strain curve for both unconfined and confined concrete regions, Modified Kent and Park concrete material model [15] is adopted. The maximum stress of the confined concrete is computed as 216.14 kg/cm^2. Unit strains corresponding to the maximum stresses for unconfined and confined concrete are assumed to be 0.002 and 0.00235, respectively. When creating the *Multilinear Isotropic Hardening* models, the stress-strain curves are assumed to be composed of straight line segments and the modulus of elasticity is calculated as the slope of the initial line segment. The modulus of elasticity for unconfined and confined concrete is computed as 160,540 kg/cm^2 and 164,200 kg/cm^2, respectively.

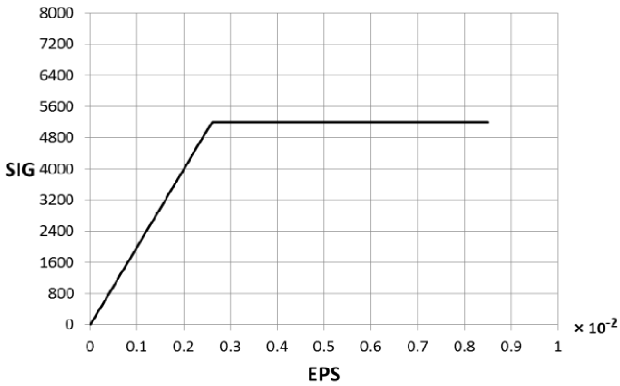

Figure 5. The ε-σ relationship of *Bilinear Isotropic Hardening Model* for the steel (SIG: kg/cm^2)

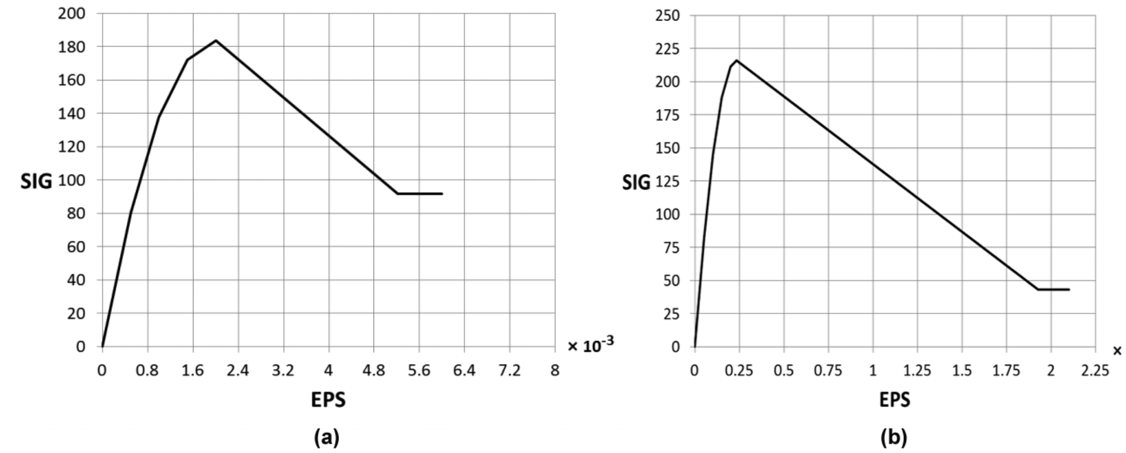

Figure 6. The ε-σ relationship of *Multilinear Isotropic Hardening Model* for: (a) Unconfined concrete (b) Confined concrete (SIG: kg/cm^2)

Since the beam-column connections did not have any stirrups, unconfined concrete material model is defined in these regions (Figure 6a). Confinement reinforcement is modeled through the confined concrete material model (Figure 6b) for the *Solid65* elements located at the confined region rather than creating separate spar elements for stirrups.

The tensile strength of concrete is assumed to be 10% of its compressive strength. The Poisson's ratios for concrete and steel are taken as 0.2 and 0.3, respectively. The control for concrete crushing in compression is prevented by inputting -1 for the concrete compressive strength in *Solid65* options. Activating concrete crushing control and using hardening models (Figure 6) concurrently is known to cause analysis and convergence errors and therefore avoided.

3.3. Analysis under lateral monotonic displacement loading

In order to investigate the monotonic behavior and crack profiles of the frame, an FE analysis is performed and the results are partially compared with the experimental data obtained under reversed cyclic loading of the frame in the laboratory. The loading used for the FE analysis is applied at the top corner of the frame as in the case of experimental loading. A displacement controlled loading is preferred and its magnitude is incrementally increased. The loading is continued until the termination of the analysis due to convergence errors. Convergence using nonlinear element types such as *Solid65* is so sensitive to the analysis and element options. The size of the load increment is initially set to an optimum value and then, let ANSYS change automatically according to the number of nonlinear iterations performed at each load step. The shear transfer coefficients for the cracks should be assumed considering both the actual behavior of the crack interface and the convergence of the analysis. In order to see the effect of different assumptions for these coefficients on the load-displacement relationship, two analyses are performed: Analysis (a) and Analysis (b). The coefficients for open and closed cracks are assumed to be 0.2 and 0.8 in Analysis (a); 0.5 and 1.0 in Analysis (b), respectively. Analysis (a) uses a more realistic assumption whereas Analysis (b) uses a more convergent assumption.

In Figure 7, the relationship between the applied forces and the measured displacements obtained from the experiment and FE analyses is shown. As expected, in Analysis (b), the frame has demonstrated a more rigid behavior. Analysis (a) with more realistic assumption has stopped earlier producing results much closer to the experimental ones.

As it is seen, both analyses have stopped at around the ultimate load capacity of the frame. Due to excessive cracking, significant convergence problems are experienced in the solution of nonlinear problems using ANSYS *Solid65* element type. Several attempts have been made to overcome these convergence problems through adjustment of load step sizes and loosening the convergence tolerances. However it was impossible to get the expected softening regime after the peak strength.

In Figure 8, the crack profile with "third" cracks obtained from the Analysis (a) at the maximum displacement (40 mm) is shown. It is observed that the cracks are concentrated at beam-column connections and distributed in accordance with the direction of the applied load.

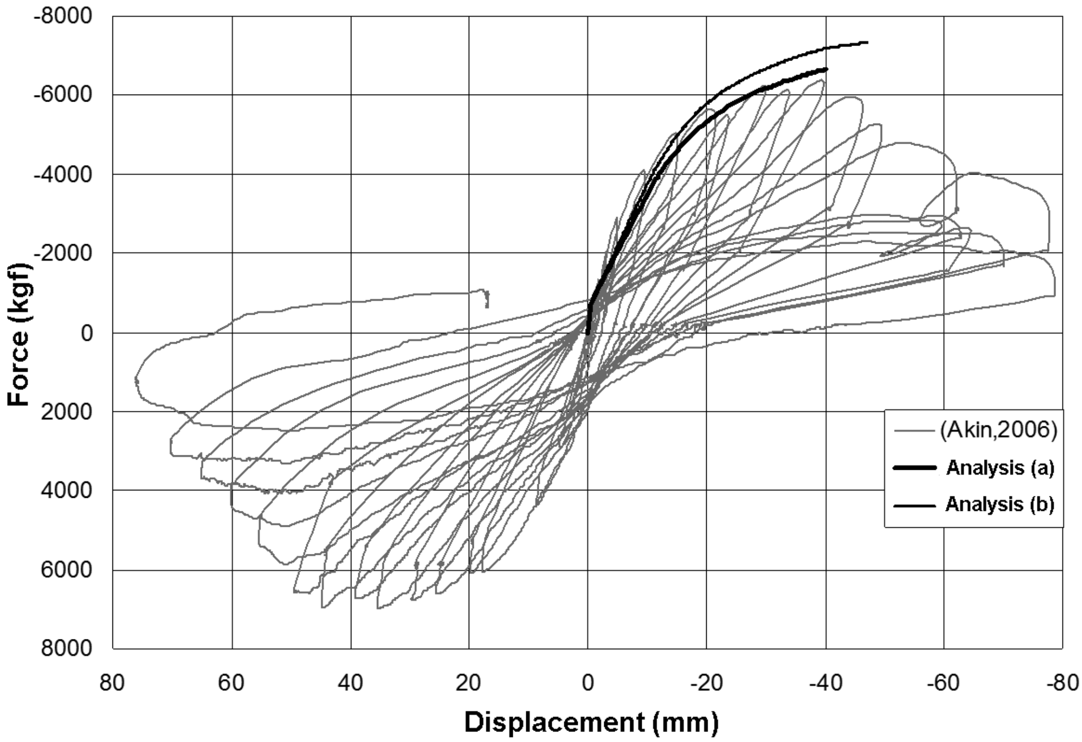

Figure 7. Comparison of analyses results under monotonic loading with the experimental data

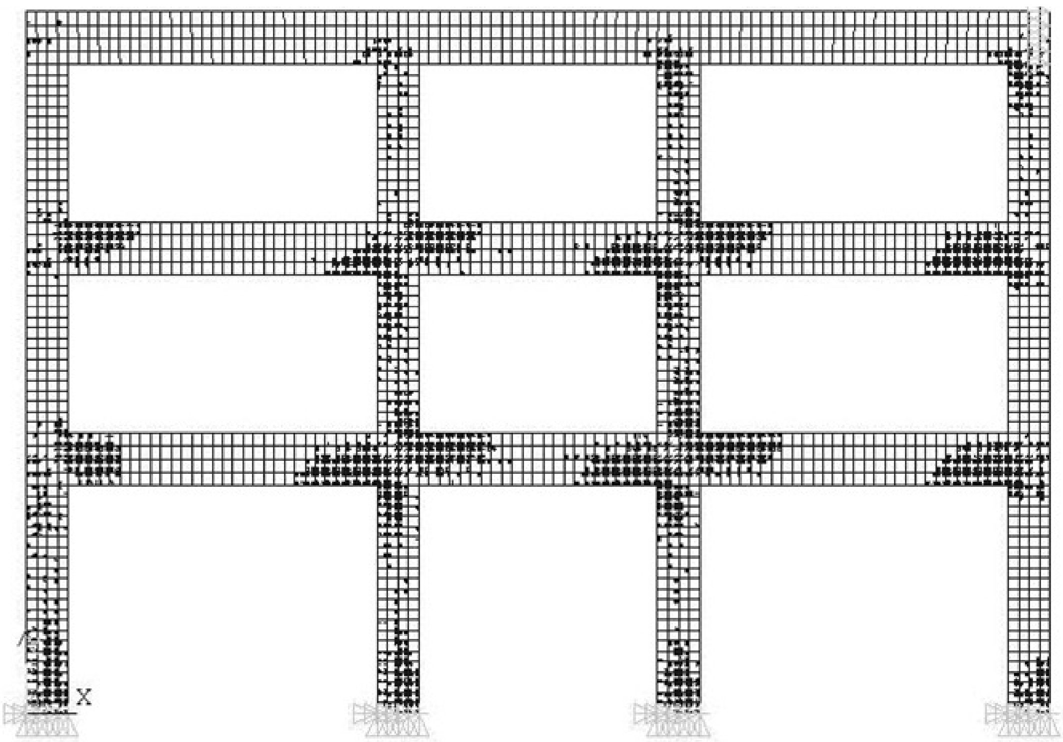

Figure 8. The crack profile with “third” cracks obtained from ‘Analysis (a)’ at maximum displacement

3.4. Analysis under lateral reversed cyclic loading

The reversed cyclic displacement loading shown in Figure 9 is applied at the nodes of the elements located at the top corners of the frame model in 9 load steps (Figure 10). The load step size is chosen to be 10 mm. For *Solid65* concrete elements, to ease the convergence of the analysis, crack shear transfer coefficients are assumed to be 0.5 and 1.0, respectively. The loading process is terminated due to non-convergence at the 9th load step with a displacement value of 29.3 mm. A total of 3889 nonlinear analysis steps are executed.

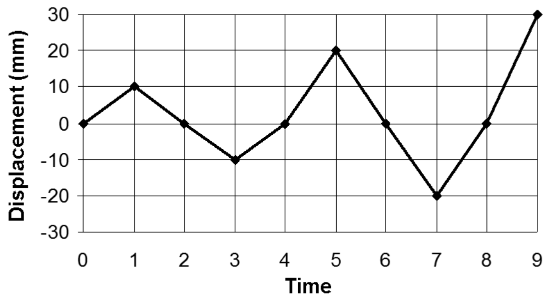

Figure 9. Reversed-cyclic displacement loading (Δu=10 mm)

In Figure 11, the experimentally obtained reversed cyclic loading vs. displacement data are compared with the analysis results. In Figure 11(a), to allow a better comparison by displaying more details, the graph is bounded by the ultimate displacement and force values obtained by the analysis. As it is seen, the analysis results are in good agreement with the experimental data. However, it is not possible to make a complete comparison for the loading-unloading curves due to the convergence problem. Similar convergency problems were experienced by Miller et al. [6].

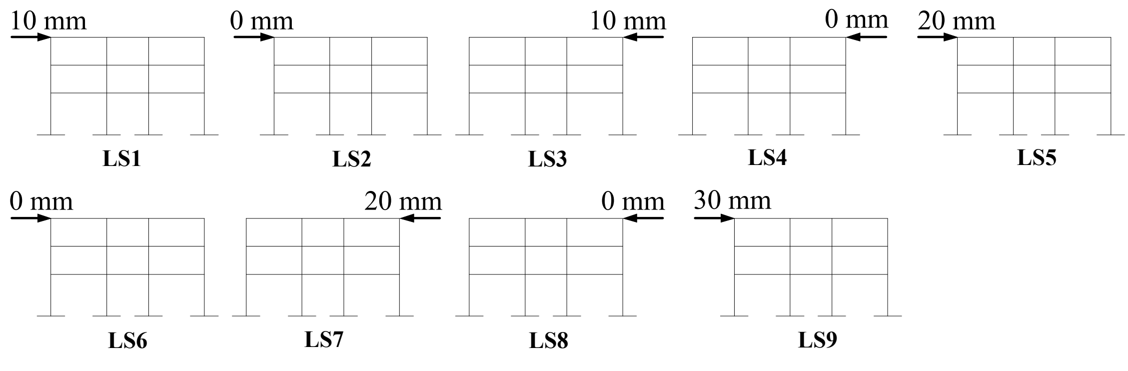

Figure 10. The displacement load steps applied at the RC frame

In Figure 12, the crack profiles with "third" cracks for different load steps are shown. In Figure 13, the experimental cracks at the maximum displacement are shown schematically. As it is seen, the number of cracks on the model in Figure 12(d) is much larger than that on the test specimen. It should be noted that cracks are displayed on the model regardless of their widths; even they are not visible to the eye. In the initial load steps, the cracks are concentrated only at the beam-column connections. As the magnitude of the load increases, the number of cracks also increases and the cracks propagate away from the connections.

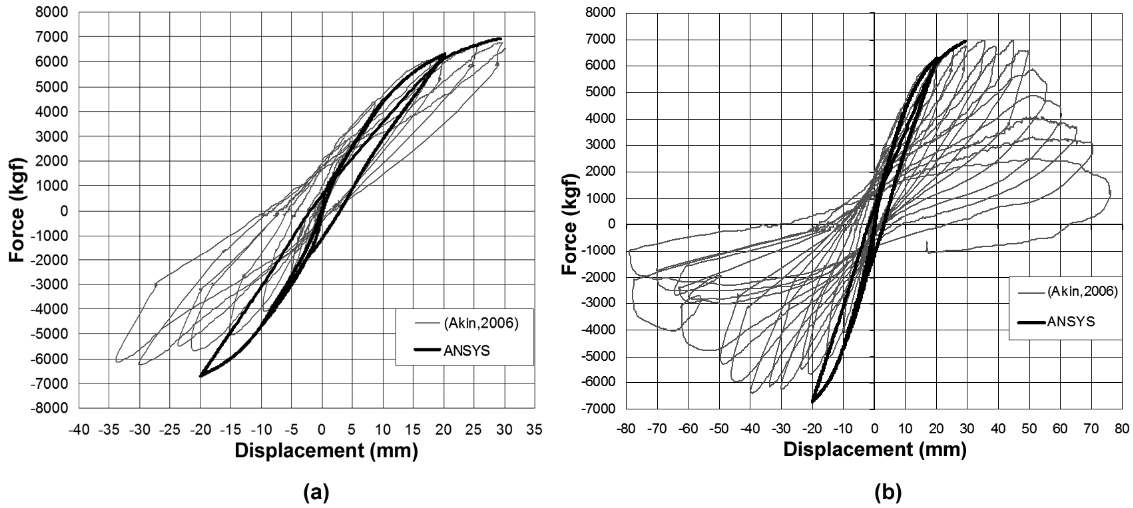

Figure 11. Comparison of analysis results with the experimental data (a) Focused view (b) General view

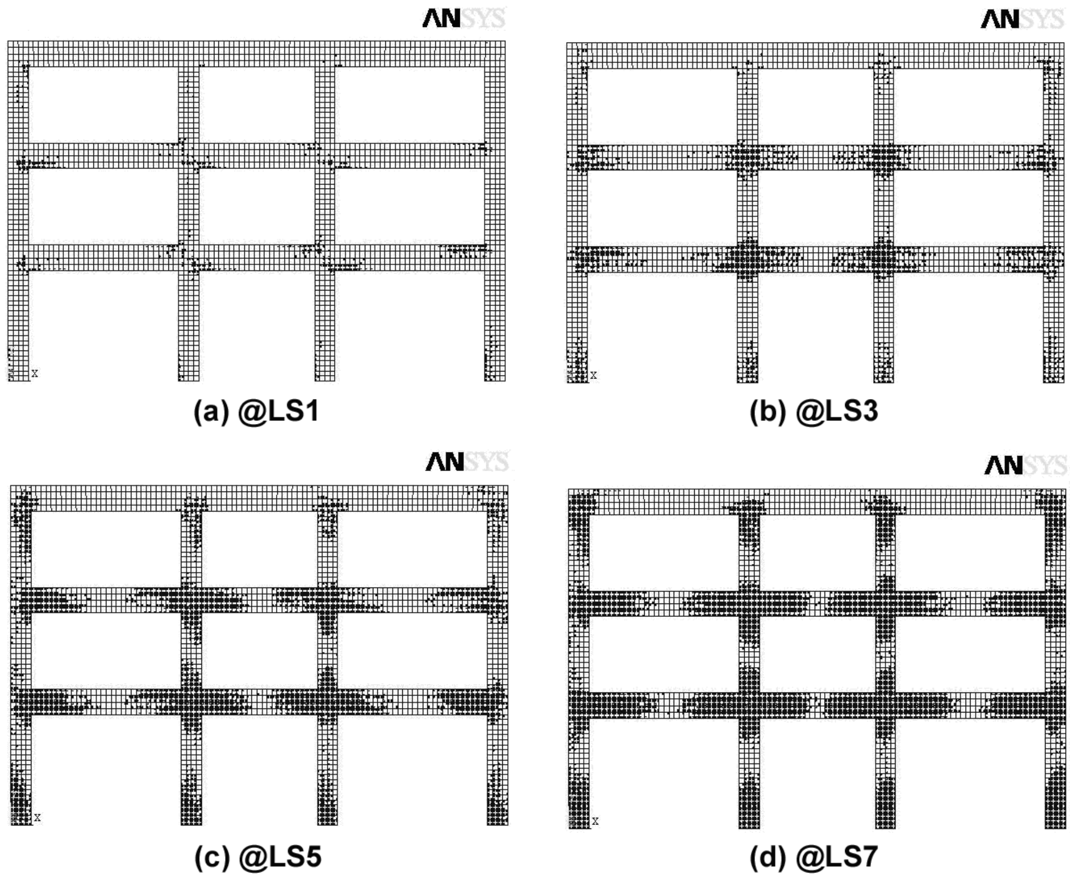

Figure 12. Calculated crack profiles at various load steps

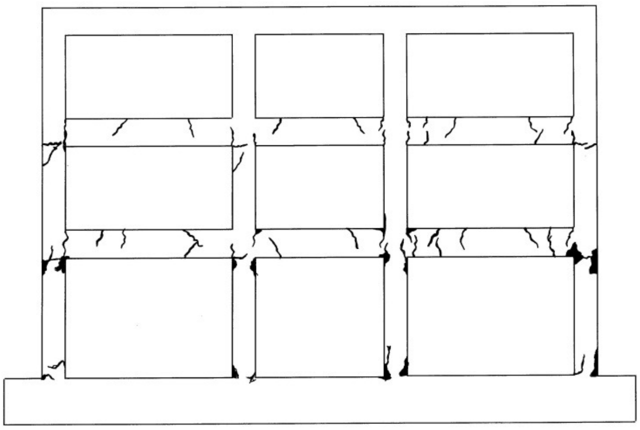

Figure 13. Observed crack profile at the maximum displacement by Akin [13]

4. CONCLUSIONS

In this work, the failure analyses of R/C structural frames under monotonic and reversed cyclic loading are carried out by using ANSYS software. The 3D FE model of an R/C frame specimen tested by Akin [13] at Selcuk University, Civil Engineering Structures laboratory is constructed by the use of *Solid65* element type which can simulate three axial cracking of concrete. The reinforcing steel can be modeled either by using discrete spar elements or by the smeared rebar option within *Solid65*. Smeared reinforcement option is preferred since it is found to be easy and reliable. The effect of various crack shear transfer coefficient assumptions for open and closed cracks is discussed. The steps of applying reversed cyclic loading onto the frame are presented. Stress-strain relationship of concrete material obtained from the confined and unconfined Kent & Park mathematical models produced quite reliable load-displacement relationship and crack profiles.

The load-displacement figures obtained from ANSYS analyses and the experimental data are compared and found to be in good agreement with each other; however, complete load-displacement relations from the analyses could not be obtained due to convergence problems near ultimate loading capacity of the frame.

The crack profiles obtained from the ANSYS model are plotted for different loading steps and compared with the experimental crack schematic figure. Although the number of cracks on the finite element model is quite larger, the crack locations compare well with the experimental ones. Unfortunately, it is not possible to define and hide the cracks invisible to the eye in ANSYS. For researchers doing experimental studies, it is proposed to carry out computer simulations of their planned experiments prior to laboratory fabrication and testing processes. This might provide significant advantages such as lesser amount of construction materials, labor and time.

5. REFERENCES

1. SAS, ANSYS 7.1 Finite element analysis system, Theoretical manual, Swanson Analysis Systems IP, Inc., 2003.

2. H.O. Köksal and B. Doran, Finite element applications using non-linear octahedral elastic and plastic relations in concrete and reinforced concrete members, Technical J. of Turkish Chamber of Civil Eng. **8**(Digest), 437-441, 1997.
3. A.F. Barbosa and G.O. Ribeiro, Analysis of reinforced concrete structures using ANSYS nonlinear concrete model, In: Idelsohn S, Onate E, Dvorkin E (Eds) Computational Mechanics: New Trends and Applications, Barcelona, Spain, 1998.
4. D. Kachlakev and T. Miller, Finite element modeling of reinforced concrete structures strengthened with FRP laminates, Technical Report, No. SPR 316, Corvallis, Oregon, USA, 2001.
5. P. Fanning, Nonlinear models of reinforced and post-tensioned concrete beams, Electronic Journal of Structural Engineering **2**, 111-119, 2001.
6. C.A. Miller, C.H. Hofmayer, Y. Wang, N. Chokshi, A.J. Murphy and Y. Kitada, Prediction of NUPEC's multi-axis loading tests of concrete shear walls, 16th International Conference on Structural Mechanics in Reactor Technology, SMiRT 16, 12-17 August, Washington, DC, USA, 2001.
7. Y. Calayir and M. Karaton, Non-linear dynamic analysis of arch dams using Drucker-Prager approach, Technical J. of Turkish Chamber of Civil Eng. **15**, 3085-3103, 2004.
8. E. Erduran and A. Yakut, Drift based damage functions for reinforced concrete columns, Computers and Structures **82**, 121-130, 2004.
9. A. Rahman, R. Hawileh and H. Tabatabai, FE model of precast hybrid beam column connection subjected to cyclic loads, 16th Engineering Mechanics Conference, ASCE, 16-18 July, University of Washington, Seattle, USA, 2003.
10. F.E. Farvashany, S.J. Foster and B.V. Rangan, Computer simulation of tests on reinforced concrete shear walls, 8th International Conference on Damage and Fracture Mechanics: Computer Aided Assessment and Control, Crete, Greece, 2004.
11. F.T. Dede, Nonlinear finite element analysis of reinforced concrete frames subjected to reversed-cyclic loading using ANSYS software, M.Sc. Thesis (in Turkish), Selcuk University Graduate School of Nat. and Appl. Sci., Konya, Turkey, 2006.
12. L. Dahmani, A. Khennane and S. Kaci, Crack identification in reinforced concrete beams using ANSYS software, Strength of Materials **42**(2), 232-240, 2010.
13. S.K. Akin, Rehabilitation of RC frames by tension only bracing, Ph.D. Thesis (in Turkish), Selcuk University Graduate School of Nat. and Appl. Sci., Konya, Turkey, 2006.
14. K.J. Willam and E.D. Warnke, Constitutive model for the triaxial behavior of concrete, In: Proceedings of International Association for Bridge and Structural Engineering: Seminar on Concrete Structures Subjected to Triaxial Stress, ISMES, Bergamo, Italy, p. 174-186, 1975.
15. R. Park, M.J.N. Priestley and W.D. Gill, Ductility of square-confined concrete columns, ASCE J Struct Div. **108**(4), 929-950, 1982.