Nonlinear Analysis of Reinforced Concrete Column with ANSYS

V. S. Pawar[1], P. M. Pawar[2]

[1]P.G. Student, Dept. Of civil Engineering, SVERI's College of Engineering Pandharpur, Maharashtra, India
[2]Professor, Dept. Of civil Engineering, SVERI's College of Engineering Pandharpur, Maharashtra, India

---

Abstract - *Different methods have been utilized in order to assess the behavior and phenomenon of typical failures like flexural, shear, torsion, buckling etc. of the Reinforced concrete structures. Typically, the behavior of reinforced concrete is studied by full-scale experimental investigations. With the invention of sophisticated numerical tools for analysis like the finite element method (FEM), it has become possible to model the complex behavior of reinforced concrete members using Finite Element modeling. In the present paper models of reinforced concrete columns subjected to axial symmetric and eccentric loading are used. Nonlinear finite element analysis is used to analyze reinforced concrete columns up to failure with FEM software ANSYS. Reinforced concrete column subjected to the axial symmetric loading, are modeled considering the frequent use in the laboratory.*

Key Words: Flexural, Torsion, Shear, Buckling, Nonlinear Finite Element Analysis, ANSYS.

1. INTRODUCTION

Experimental analysis is widely carried out to study individual component members and the concrete strength under various loading conditions. This method provides the actual behavior of the structure. But it is time consuming and expensive. Finite element analysis is also used to analyze these structural components. Finite Element Analysis (FEA) is a method used for the evaluation of structures, providing an accurate prediction of the component's response subjected to various structural loads. The use of FEA has been the preferred method to study the behavior of concrete as it is much faster than the experimental method and is cost effective. With the invention of sophisticated numerical tools for analysis like the finite element method (FEM), it has become possible to model the complex behavior of reinforced concrete columns using Finite Element modeling. Finite element analysis can also be used to model the behavior numerically to confirm these calculations, as well as to provide a valuable supplement to the laboratory investigations, particularly in parametric studies.

Barbosa et al. [1] considered the practical application of nonlinear models in the analysis of reinforced concrete structures and the consequences of small changes in modeling. The best results were obtained from the elastoplastic-perfectly plastic, work-hardening models that reached ultimate loads, very close to the predicted values.

Dahmani et al. [2] conducted an investigation into the applicability of ANSYS software for analysis and prediction of crack patterns in RC beams and the advantage of performing numerical simulation instead of experimental tests. For this purpose, different phases of the behavior of the FE model of an RC beam was studied from initial cracking to failure of the beam.

Wolanski [3] in his thesis work, studied reinforced and pre-stressed concrete beams using Finite Element Analysis (FEA) to understand their load-deformation response. The results were compared to experimental data. Characteristic points on the load-deformation curve predicted using FEA were then compared to theoretical results. The nonlinear analysis of the model yielded results that compared well to the calculated values.

Kachlakev et al. [4] finite element method (FEM) models are developed to simulate the behavior of four full-size beams from linear through nonlinear response and up to failure, using the ANSYS program (ANSYS 1998). Comparisons are made for load-strain plots at selected locations on the beams; load-deflection plots at mid-span; first cracking loads; loads at failure; and crack patterns at failure.

In this paper, the nonlinear material model of concrete in commercially available finite element software ANSYS v15 is used to evaluate the ultimate load, Load mid-span displacement relationship and cracks development of reinforced concrete columns.

Elements, material models and techniques of nonlinear analysis suggested by past researchers who validated their results with experiments, are used, so that results are in the range of expectations according to theory and need not to be validated.

2. FINITE ELEMENT MODELLING

2.1 Element Types

Concrete: The Solid65 element is used to model the concrete. This element has eight nodes with three degrees of freedom at each node – translations in the nodal x, y, and z directions. This element is capable of plastic deformation, cracking in three orthogonal directions, and crushing. A schematic of the element is shown in Fig- 1.

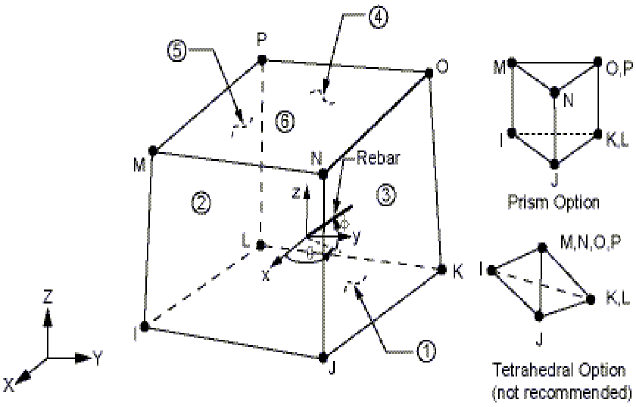


Fig-1: Solid 65 element

Steel Reinforcement: A Link180 element is used to model steel reinforcement. This element is a 3D spar element and it has two nodes with three degrees of freedom – translations in the nodal x, y, and z directions. This element is also capable of plastic deformation. This element is shown in Fig- 2.

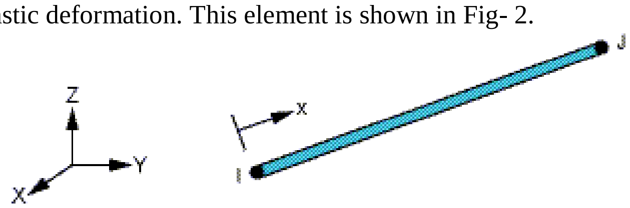


Fig-2: Link 180 element

Steel Plates and Supports: Steel plate and supports are modeled using element called Solid185. This element is defined by eight nodes having three degrees of freedom at each node translations in the nodal x, y, and z directions. The element is capable of plasticity, hyper elasticity, stress stiffening, creep, large deflection, and large strain capabilities.Solid185 is available in two forms:
Homogeneous Structural Solid (default); and
Layered Structural Solid.
Homogeneous Structural Solid with simplified enhanced strain formulation is used to model steel plate for application of load. This element is shown in Fig-3.

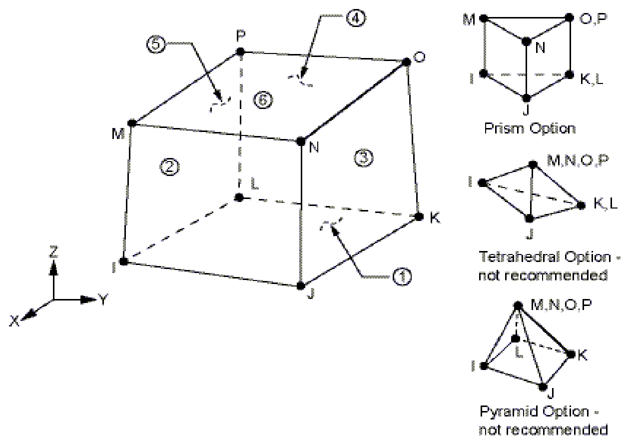


Fig-3: Solid185 Element (Homogeneous Structural Solid) in ANSYS

2.2 Material Properties

Concrete: Concrete is a quasi-brittle material and has highly nonlinear and ductile stress strain relationship. The nonlinear behaviour attributed to the formation and gradual growth of micro cracks under loading. The tensile strength of concrete is typically 8-15% of the compressive strength. Fig-4 shows a typical stress-strain curve for normal weight concrete [5]. In compression, the stress-strain curve for concrete is linearly elastic up to about 30% of the maximum compressive strength. Above this point, the stress increases gradually up to the maximum compressive strength. After it reaches the maximum compressive strength σ_{cu}, the curve descends into a softening region, and eventually crushing failure occurs at an ultimate strain ε_{cu}. In tension, the stress-strain curve for concrete is approximately linearly elastic up to the maximum tensile strength. After this point, the concrete cracks and the strength decreases gradually to zero [5].

Various mathematical models are available to approximate this nonlinear behaviour. In this paper, the compressive uniaxial stress-strain relationship for the concrete is obtained using modified Hognestad piecewise elastic model [6]. Following equations are used to compute the multilinear isotropic stress-strain curve for the concrete (Fig-5).

$$E_c = 5000\sqrt{f_{ck}} \quad (1)$$

$$f = \frac{E_c \varepsilon}{1 + \left(\frac{\varepsilon}{\varepsilon_0}\right)^2} \quad (2)$$

$$\varepsilon_0 = \frac{2 f_{ck}}{E_c} \quad (3)$$

$$E_c = \frac{f}{\varepsilon} \quad (4)$$

Where:
f = stress at any strain ε, MPa
ε = strain at stress f
ε_0 = strain at the ultimate compressive strength f_{ck}.

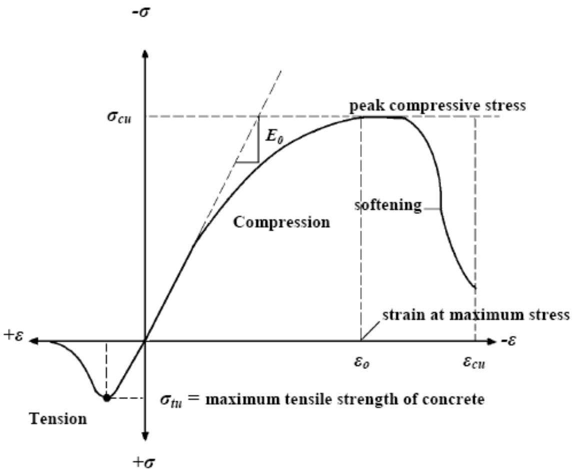


Fig-4: Typical uniaxial compressive and tensile stress-strain curve for concrete [5]

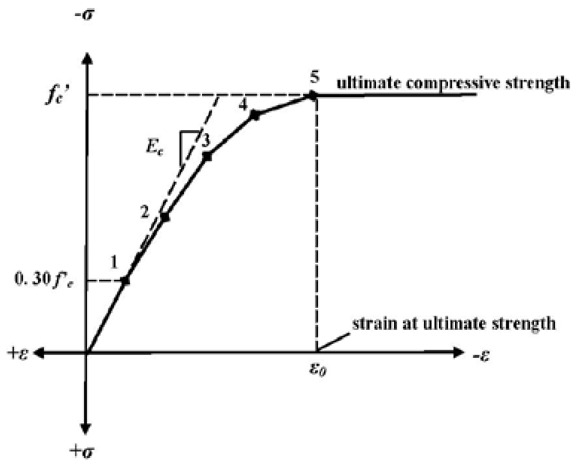


Fig-5: Typical uniaxial compressive and tensile stress-strain curve for concrete [5]

The model is capable of predicting failure for concrete materials (Fig-6). Both cracking and crushing failure modes are accounted for. The two input strength parameters i.e., ultimate uniaxial tensile and compressive strengths are needed to define a failure surface for the concrete. Consequently, a criterion for failure of the concrete due to a multiaxial stress state can be calculated [7].

In a concrete element, cracking occurs when the principal tensile stress in any direction lies outside the failure surface. After cracking, the elastic modulus of the concrete element is set to zero in the direction parallel to the principal tensile stress direction.

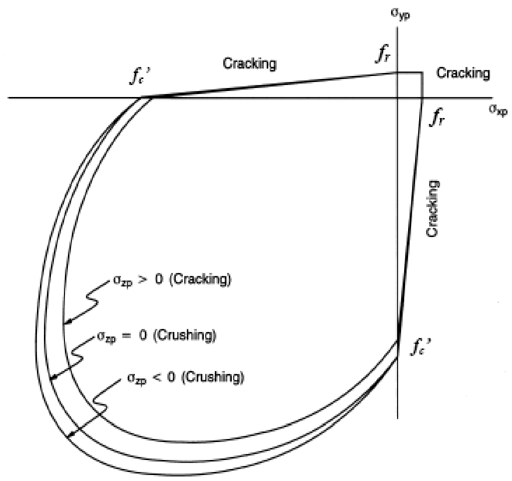


Fig-6: Failure Surface for concrete

Crushing occurs when all principal stresses are compressive and lies outside the failure surface; subsequently, the elastic modulus is set to zero in all directions, and the element effectively disappears. During the study of past researchers [3], [4], it was found that if the crushing capability of the concrete is turned on, the finite element beam models fail prematurely. Therefore, in this study, the crushing capability was turned off and cracking of the concrete controlled the failure of the finite element models.

For Implementation of the Willam and Warnke [7], material model in ANSYS, nine constants are defined as : Shear transfer coefficients for an open crack; Shear transfer coefficients for a closed crack; Uniaxial tensile cracking stress; Uni-axial crushing stress (positive); Biaxial crushing stress (positive); Ambient hydrostatic stress state for use with constants 7 and 8; Biaxial crushing stress (positive) under the ambient hydrostatic stress state (constant 6); Uniaxial crushing stress (positive) under the ambient hydrostatic stress state (constant 6); Stiffness multiplier for cracked tensile condition.

Steel Reinforcement: The mechanical behaviour of reinforcing steel bar is assumed to be elastic bilinear under monotonic tension. The steel bar initially exhibits linear elastic portion followed by a yield platue, strain hardening and then stress drops till fracture occurs. The behaviour of steel bar is same in compression and tension loading, as shown in Fig- 7. For incorporating steel material model, the essential inputs are modulus of elasticity, tangent modulus and the yield strength. Here work hardening part is added for the steel properties for the stability of the FE model of the beam.

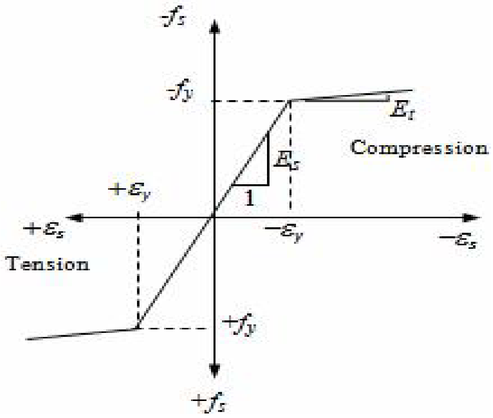


Fig-7: Strain curve for the steel reinforcement

2.3 Finite Element Modeling of Steel Reinforcement

There are three techniques that exist to model steel reinforcement in finite element models for reinforced concrete: the discrete model, the embedded model, and the smeared model [8]. In the work presented in this paper, discrete modeling technique is used for modeling the reinforcement. The reinforcement in the discrete model (Fig-8) uses bar or beam elements that are connected to concrete mesh nodes. Therefore, the concrete and the reinforcement mesh share the same nodes and concrete occupies the same regions occupied by the reinforcement.

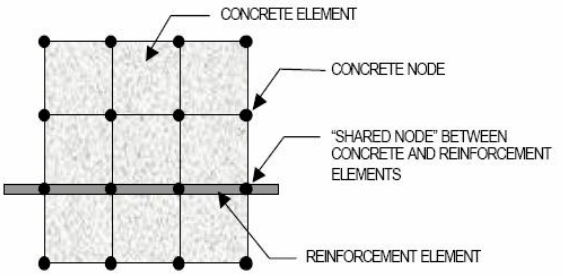


Fig- 8: Discrete Models for Reinforcement [8]

3. ANALYSIS OF RC BEAMS USING ANSYS

3.1 Modeling and Meshing

The beam is modeled by creating volumes. To obtain good results from the Solid65 element, rectangular mesh is used. The individual elements of reinforcement are created in the modeling through the nodes created by the mesh of the concrete volume. The command 'merge items' merges separate entities that have the same location. These items will then be merged into single entities.

Displacement boundary conditions are needed to constrain the model to get a unique solution. To ensure that the model acts the same way as the experimental beam, boundary conditions need to be applied at the supports and loadings. The support is modelled in such a way that a hinge and roller is created. The force applied is applied across the entire nodal line.

3.2 Nonlinear Analysis

In nonlinear analysis, the total load applied to a finite element model is divided into a series of load increments called load steps. At the completion of each incremental solution, the stiffness matrix of the model is adjusted to reflect nonlinear changes in structural stiffness before proceeding to the next load increment. The ANSYS program uses Newton-Raphson equilibrium iterations for updating the model stiffness. Newton-Raphson equilibrium iterations provide convergence at the end of each load increment within tolerance limits.

3.3 Load Stepping and Failure Definition for FE Models

For the nonlinear analysis, automatic time stepping in the ANSYS program predicts and controls load step sizes. Based on the previous solution history and the physics of the models, if the convergence behavior is smooth, automatic time stepping increases the load increment up to a selected maximum load step size. The maximum and minimum load step sizes are required inputs for the automatic time stepping.

Failure for each of the models is defined when the solution for a 1 N load increment still does not converge. The program then gives a message specifying that the models have a significantly large deflection, exceeding the displacement limitation of the ANSYS program.

4. NONLINEAR FINITE ELEMENT ANALYSIS OF REINFORCED CONCRETE COLUMNS

4.1 Problem Statement

The reinforced column having cross section 250mm both side and height of 1250mm reinforced with 4-12 mm bars with 6 mm stirrups at 120 mm spacing and concrete is of M 25 grade. Cover of 25mm is provided in all sides. Details are shown in the Fig-9.

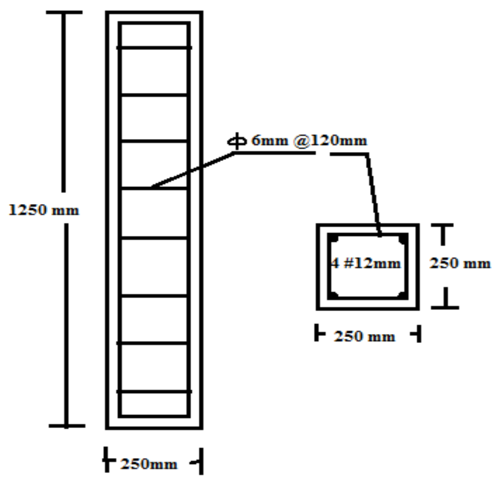
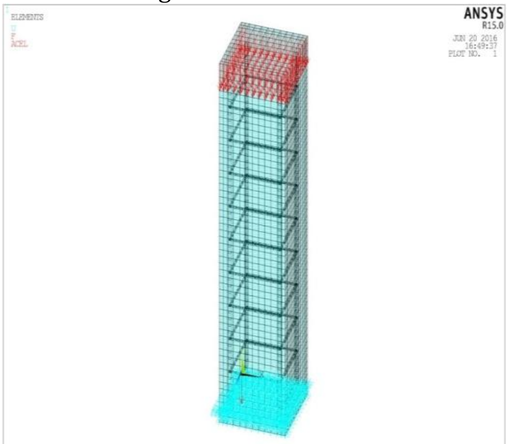


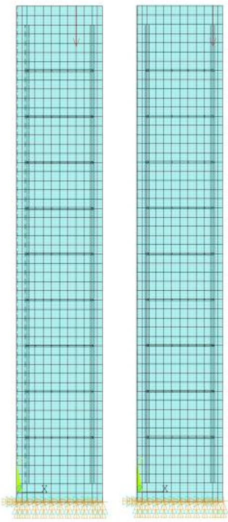
Fig-9: Details of reinforced concrete column

Total two columns are analyzed in this present work. The column which is shown in Fig-9 is analyzed with axial symmetrically distributed loading and eccentric loading with 50mm and 100mm eccentricities. Another column called as long column having same configuration but its height is about 4800mm and is subjected to eccentric load with eccentricity of 50mm.

For present analysis, RC columns are modeled in ANSYS 15 using GUI with the details given above. The FE model in ANSYS is shown in Fig-10.



(a)



(b)

Fig-10: FE models of columns (a) Axial symmetric loading, (b) Eccentric loading of 50mm and 100mm.

Support is modeled as hinged and fixed support at the bottom of the column for axial and eccentric columns respectively by constraining the displacement in the relevant directions. Load is given in increments.

4.2 Results and Discussion

Results and Discussions: Plots of load versus mid-span deflection for uniform axial loaded column from FEA is shown in Chart-1.

Load mid-span deflection plot

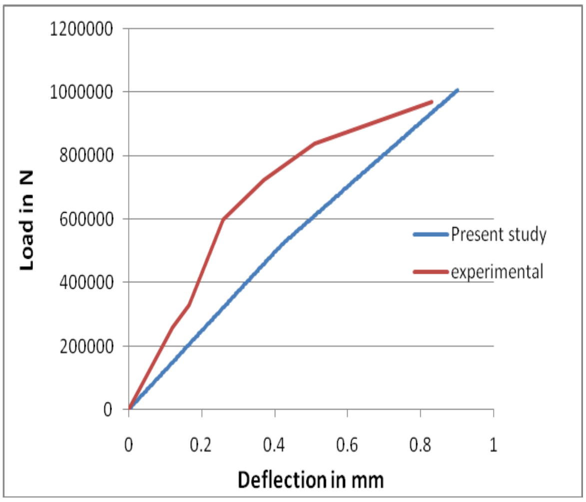


Chart-1: Load Mid-span deflection plot for column model

Chart-1 shows the load mid-span deflection plot and shows the correlation of experimental and FEA analysis with ANSYS.

Vertical deflection in columns

In the axial column vertical deflection is more at the free end where load is applied as shown in Fig-11.

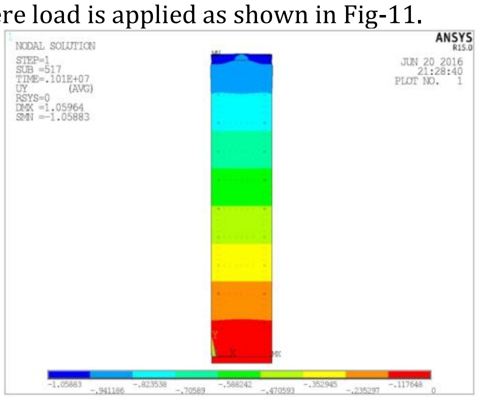


Fig-11: Axial column

As eccentricity increases the maximum value shifted towords the eccentricity and also the maximmum value increases because of the bending effect.

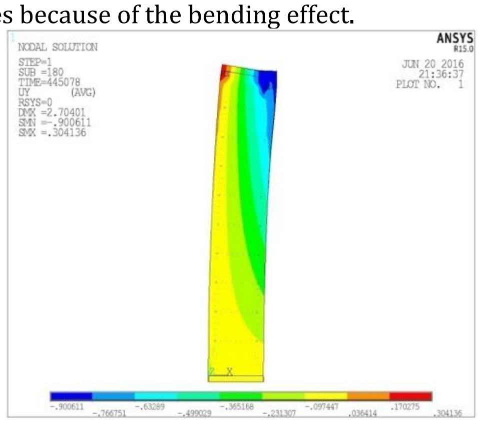


Fig-12: Column with 50mm eccentricity

Fig-13 and 14 shows the same effects of the eccentricity.

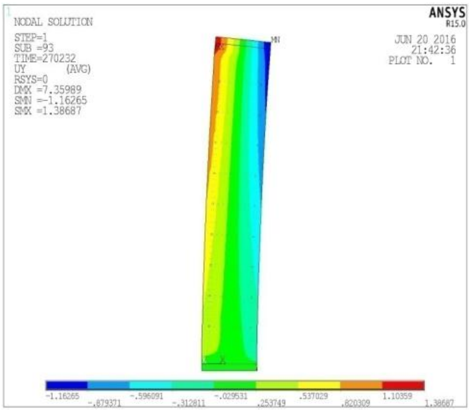


Fig-13: Column with 100mm eccentricity

Horizontal Deflection in Columns

Column subjected to the axial loading shows very less horizontal deflection and this is due to deformation of concrete due shear failure at top and extreme bottom contact surface.

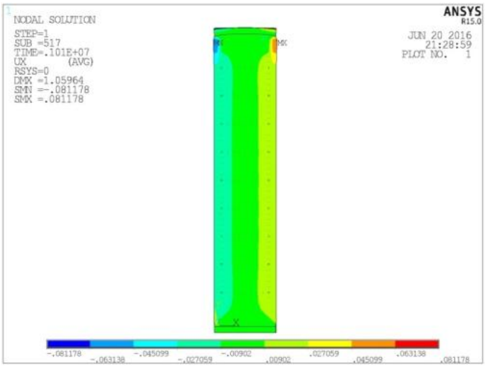


Fig-14: Axial column

As eccentricity increases the horizontal deflection increases as shown in the Fig-15 and 16. Tension increases at the opposite side of the eccentricity.

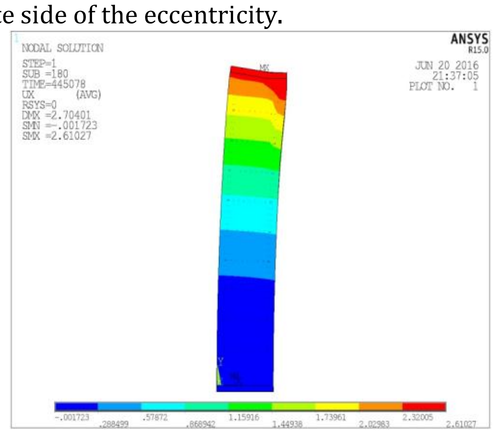


Fig-15: Column with 50mm eccentricity

Also from Fig.15 and 17 shows that for the same eccentricity long column posses more horizontal deflection than the short column. This is due to the slenderness effect causing buckling in the long column and hence failure starting with large horizontal deflection.

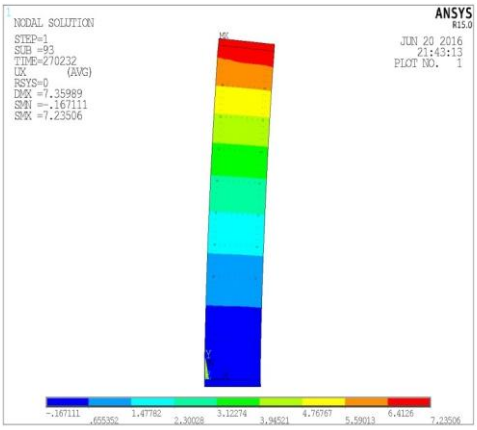


Fig-16: Column with 100mm eccentricity

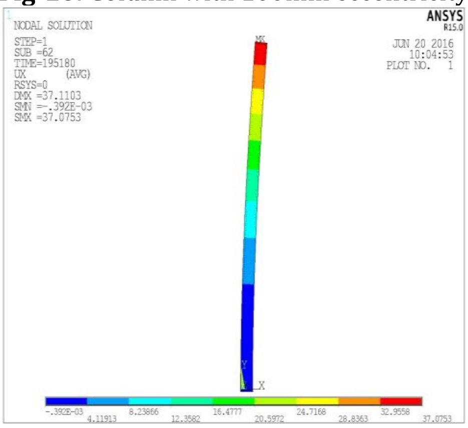


Fig-17: Long column with 50mm eccentricity

Cracks in concrete of Columns

Concrete is a brittle material and hence failure of column starts with failure of concrete. Fig-18 shows the crack distribution in concrete with axial and eccentric loaded column. We can see in axial column concrete posses diagonal tensile cracks as failed in shear at top of the column just below the load.

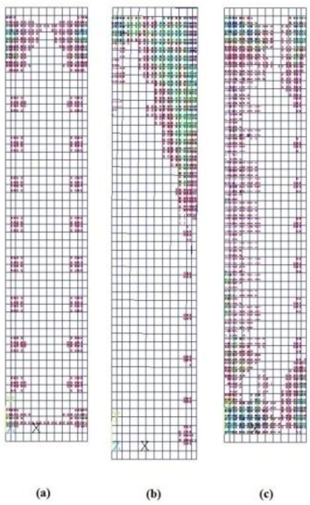


Fig-18: Cracks in axial and eccentric columns

As eccentric loads are applied shearing zone shifted towords load. Further tensile cracks occur on the opposite side and column tends to fails with buckling.
Fig-19 shows the slenderness effect on the concrete failure in short and long column with same eccentricites. Short column shows more diagonal tensile cracks i.e. shear failure and tensile cracks will not occur on opposite side. On the other hand in the long column shear failure diminished because concrete in the column tends fail in tensile side.

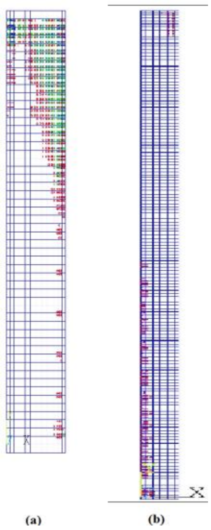


Fig-19: Cracks in short and long columns

5. CONCLUSIONS

The failure mechanisms of axial and eccentric reinforced concrete columns are modelled quite well using FEA, and are distinct and very close to the expectations. Finite element models of reinforced concrete columns, constructed in ANSYS 15 using the dedicated concrete elements have accurately captured the nonlinear combined axial and bending response of these systems up to failure. The analysis procedure used in this paper suggested by past researchers and various output plots constructed by FEA have provided a deeper insight and validation for parametric study for future application of finite element software for the non-linear analysis of RC columns. Based on the analyses carried out on the RC columns using ANSYS, it is found that results are more sensitive to mesh size, materials properties, load increments, etc.

ACKNOWLEDGEMENT

This work is supported by all experienced staff and Civil Engineering Department of College of Engineering Pandharpur, Maharashtra, India, for providing us valuable guidance and ANSYS 15.0

REFERENCES

[1] Antonio F. Barbosa and Gabriel O. Ribeiro, "Analysis Of Reinforced Concrete Structures Using Ansys Nonlinear Concrete Model", Computational Mechanics, New Trends And Applications, Barcelona, Spain (1998).
[2] L. Dahmani, A. Khennane, and S.Kaci, "Crack Identification in Rein-forced Concrete Beams using ANSYS Software," Strength of Materials, Vol. 42, No. 2, pp. 232–240, 2010.
[3] A.J. Wolanski, "Flexural Behavior of reinforced and Prestressed Concrete Beams Using Finite Element Analysis", Master of Science The-sis, Marquette University, Wisconsin , 2004.
[4] D. Kachlakev, T. Miller, S. Yim, "Finite Element Modeling of Rein-forced Concrete Structures Strengthened with FRP Laminates", Final Report SPR 316, Oregon Department of Transportation , 2001.
[5] M.Y.H. Bangash, "Concrete and Concrete Structures: Numerical Modeling and Applications", Elsevier Science Publishers Ltd., London, England. 1989.
[6] J.G. MacGregor, "Reinforced Concrete Mechanics and Design", Prentice-Hall, Inc., Englewood Cliffs, NJ., 1992.
[7] K.J. Willam, and E. P. Warnke, "Constitutive Model for Triaxial Behavior of Concrete," Seminar on Concrete Structures Subjected to Triaxial Stresses, International Association of Bridge and Structural Engineering Conference, Bergamo, Italy, 174. (1974),
[8] F.A. Tavarez, "Simulation of Behavior of Composite Grid Rein-forced Concrete Beams Using Explicit Finite Element Methods", Master of Science Thesis, University of Wisconsin–Madison, Madison, Wisconsin, 2001.