Study of Behavior of Reinforced Concrete Beam Using ANSYS

Neha S. Badiger
Assistant Professor
Department Of Civil Engineering
Shree L. R. Tiwari College Of Engineering, Mira Road (E)
Neha.Badiger@Gmail.Com

Abstract— Concrete structural components such as beams, columns, walls exist in various buildings and bridges. Understanding the response of these components of structures during loading is crucial for the development of an efficient and safe structure. Recently Finite Element Analysis (FEA) is also used to analyze these structural components. In this paper, four point bending analysis is carried out using reinforced concrete beam. The results of the beam with respect to mesh density, varying depths, use of steel cushions for support and loading points, effect of shear reinforcement on flexure behaviour, impact of tension reinforcement on behaviour of the beam are analyzed and discussed. Finite element software ANSYS 13.0 is used for modeling and analysis by conducting non linear static analysis.

Index Terms*— Material nonlinearity, Finite element analysis, Convergence, Steel cushion, Varying depths, ANSYS.*

*****

I. INTRODUCTION

Concrete structural components such as beams, columns, walls exist in various buildings and bridges. Understanding the response of these components of structures during loading is crucial for the development of ancient and safe structure. Different methods are used to study the response of these structural components. 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. Recently Finite Element Analysis (FEA) is also used to analyze these structural components.

II. FINITE ELEMENT ANALYSIS

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 elective. 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 beams using Finite Element modeling. Finite element method is a numerical analysis method that divides the structural element into smaller parts and then simulates static loading conditions to evaluate the response of concrete. The use of this technique is increasing because of enormous advancement of engineering and computer knowledge. This method respond well to non linear analysis as each component possesses different stress-strain behavior. The response of each element is expressed in terms of a finite number of degrees of freedom characterized as the value of an unknown function at a set of nodal points. In reality most of the problems are non-linear in nature. Hence non linear analysis is an effective tool to obtain exact solution [1].

III. PROBLEM CONSIDERED FOR THE STUDY

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 beams using Finite Element modeling.

A. Geometry of the beam

The geometry of the full size beam is 4000mm x 250mm x 450mm. The span between the two supports is 3840mm. Beam is simply supported by providing roller support on both the sides. Two point loads are applied at the midspan of the beam. M30 grade concrete and Fe415 steel is used. The details of the RC beam are as shown in Fig1

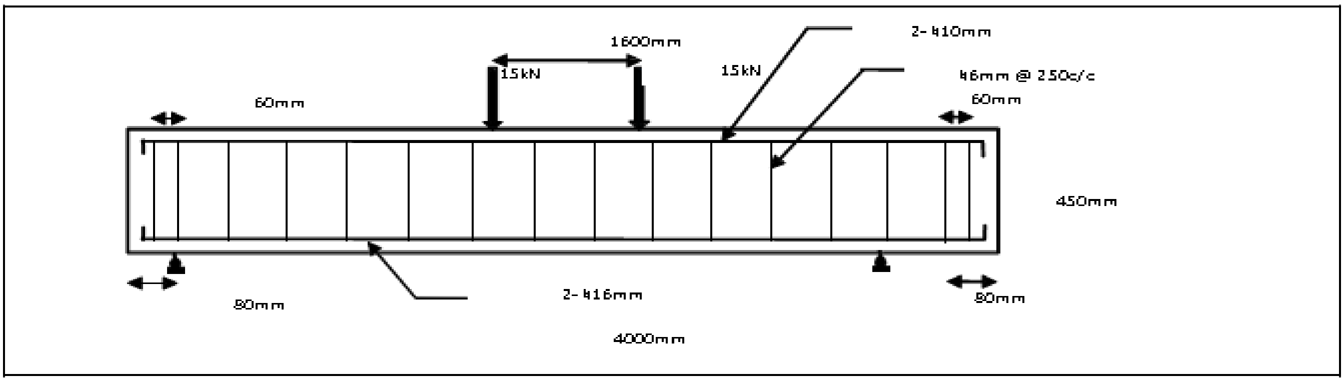

Fig 1: Beam Considered For the Study

B. Material Properties

➢ ***Steel Reinforcement***

Steel reinforcement in RC beam is of grade Fe415. The steel for the finite element models has been assumed to be an elastic-perfectly plastic material and identical in tension and compression. Poisson's ratio of 0.3 has been used for the steel reinforcement in this study. Elastic modulus equal to 200,000 MPa and Poisson's ratio of 0.3 has been used for all the reinforcing bars. Steel plates were provided at support locations and at the loading point in the finite element models (as in the actual beams) to provide a more even stress distribution over the support and loading areas. Same elastic modulus equal to 200,000 MPa and Poisson's ratio of 0.3 were used for the plates. The steel plates were assumed to be linear elastic materials. Tangent modulus of 20N/mm2 is used for reinforcement to avoid loss of stability upon yielding.[9]

Table: 1 Material property for the Beam Models

Material Model Number	**Element Type**	**Material Properties**		
1	Solid65	**Linear Isotropic**		
		EX	30000MPa	
		PRXY	0.2	
		Multilinear Isotropic		
			Strain	Stress (MPa)
		Point 1	0.00032	9
		Point 2	0.0006	15.28
		Point 3	0.0009	21.08
		Point 4	0.0012	25.27
		Point 5	0.0015	27.96
		Point 6	0.0018	29.42
		Point 7	0.0021	29.96
		Point 8	0.00219	30
2	Solid185	**Linear Isotropic**		
		EX	200000MPa	
		PRXY	0.3	
3	Link180	**Linear Isotropic**		
		EX	200000MPa	
		PRXY	0.3	
		Bilinear Isotropic		
		Yield stress	415MPa	
		Tangent Modulus	20Mpa	

- ***Concrete Properties***

Modeling an element for the behavior of concrete is a challenging task. Concrete is a brittle material and has exhibit different behavior in compression and tension. The tensile strength of concrete is typically 8-15% of the compressive strength [9]. The modulus of elasticity of concrete is calculated as 27386MPa as per IS 456:2000. Poisson's ratio is 0.2. The shear transfer coefficient for open crack and closed crack are 0.3 and 0.95 respectively. Uniaxial tensile cracking stress is obtained using IS 456:2000 and is 3.834MPa. Concrete material properties are shown in table 1 above.

- ***Compressive Uniaxial stress strain relationship***

The ANSYS program requires the uniaxial stress-strain relationship for concrete in Compression. The Solid65 element requires linear isotropic and multi-linear isotropic material properties to properly model concrete. The multi-linear isotropic material uses the Von-Misses failure criterion to define the failure of the concrete. [8] Simplified stress strain relationship for concrete in compression is obtained and is shown in Fig 2.

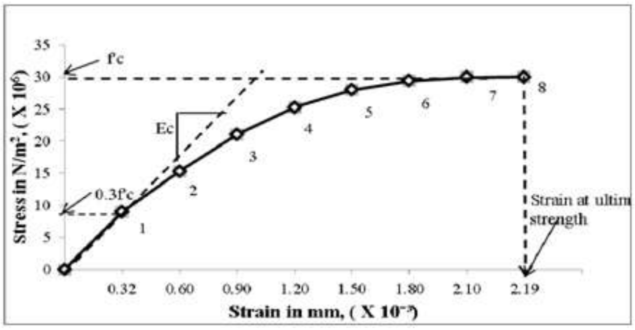

Fig 2: Simplified uniaxial stress-strain curve for concrete in compression

IV. Elements used for Modeling

Concrete beam is modeled using eight node Solid65 element which has three degrees of freedom at each node. For modeling steel reinforcement, Link180 spar element with three degrees of freedom at each node is used.

Supports and loading points are modeled using eight noded Solid180 elements.

A. *Modeling of Reinforced Concrete Beam*

The beam, plates and the support are modeled as volumes. Since a quarter of the beam is modeled, the beam is 2000mm long is having a cross section of 125mm x 450mm. The dimensions of the concrete volume are as shown in the table 2.

Table: 2 Dimensions for Concrete, Steel Plate, and Steel Support Volumes

ANSYS	Concrete (mm)		Steel Plate mm)		Steel Support (mm)	
X1,X2 X-coordinates	0	2000	1160	1240	40	120
Y1,Y2 Y-coordinates	0	450	450	475	0	-25
Z1,Z2 Z-coordinates	0	125	0	125	0	125

Loading ***and Boundary Condition***

Boundary conditions are required to get a proper solution for the model. Because a quarter of the entire beam is used for the model, planes of symmetry are required at the internal faces. The symmetric boundary condition is set first. The beam model being used is symmetric about two planes i.e. X and Z plane. [6] A quarter of the full beam was used for modeling by taking advantage of the symmetry of the beam and loadings. Planes of symmetry were required at the internal faces. At a plane of symmetry, the displacement in the direction perpendicular to that plane was held at zero. The load P applied at the steel plate is applied across the entire center line nodes of the plate. The force applied at each node on the plate is one twelfth of the actual load applied.

analysis type

A nonlinear structural analysis is performed to study the nonlinear material behavior of concrete beam. ANSYS13.0 employs "Newton-Raphson" method to solve nonlinear problems. In order to predict the nonlinear material behavior, the load is sub divided into series of load increments. The load increment can be applied over several load steps. The number of load steps required for the study is given and the time for each load step is mentioned. During the initiation of concrete crack, the steel yielding stage and at the ultimate stage where large numbers of cracks occurs, the loads are applied gradually with smaller load increments. Failure of the model is identified where the solution fails to converge even with very low load increment.

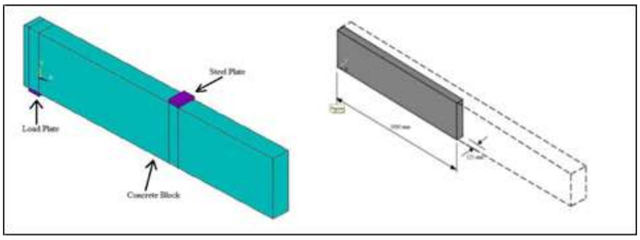

Fig 5: Volumes Created in ANSYS Quarter beam model

V. RESULTS AND DISCUSSION

Comparison of the Load-Deflection Curve for different depths

A parametric study is performed on the depth of the beam to study the behavior of the beam. Load deflection Curve for different depths of beam are done and the load at first crack is obtained. The depths adopted are 250mm, 350mm, 450mm and 500mm.

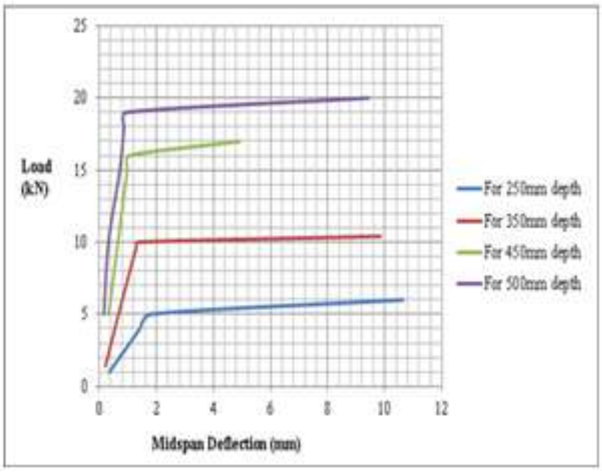

Fig 6: Comparison of Load-Deflection curves for all the depths

Comparison of Load-Deflection curves for all the depths are shown in fig 6 above. It is observed that as the depth of the beam increases, fracture instability is affected. Load carrying capacity of the beam increases with increasing depth but the deflection also increases. For 500mm depth, though the load at first crack is 20kN, the deflection is more. It is about 9.439mm.Also for beam of 250mm depth, load at first crack is early at 6kN and the deflection is also too high about 10.635mm. For beam of 350mm depth, load at first crack is about 10.4kN which is moderate but the deflection is high again. For beam of 450mm depth, load at first crack is about 17kN which is moderate and deflection is also not too high or too low. It is about 4.93mm.

VI. CONCLUSIONS

In this study, the behavior of reinforced concrete beam is analyzed using finite element method. A control beam is analyzed using a specific set of control data and is then compared to the succeeding models by changing the parameters. The parameters used to complete this study are varying depths, steel percentage, steel cushion and shear reinforcement. After compiling and analyzing the results from each test, the following conclusions can be

1 Reinforced concrete beam can be modeled and analyzed using ANSYS 13.0 software and obtain accurate results.
2. Deflections and stresses at the centerline along with initial and progressive cracking of the finite element model compare well with the manual calculations obtained for a reinforced concrete beam.
3. As the depths of the beams are increased, the load carrying capacity increases but the deflection. For smaller depths, load carrying capacity is low and deflection is also very large.
4. It is observed that by varying the tension steel, the initial cracking behavior is not affected. But it has more impact in the post cracking stage of the beam. The ultimate capacity of beam can be varied by varying the steel percentage.
5. By removing steel plate at support and loading point, stress concentration takes place. Also the beam without steel plate shows more cracks than the beam with steel plate. Hence for more accurate analysis, steel cushion has to be included in modeling.

REFERENCES

[1] E.Chandrasekaran and R.Santhakumar, "Analysis of Retrofitted Reinforced Concrete Shear Beams using Carbon Fiber Composites" Electronic Journal of Structural Engineering, April 2004.
[2] Hamid Saadatmanish, Mohammad R Ehsani and Tom Norris, "Shear and Flexural Strengthening of RC beams with Carbon Fiber Sheets" Journal of Structural Engineering, July 1997.
[3] P. Fanning, "Nonlinear Models of Reinforced and Post-tensioned Concrete Beams", Electronic Journal of Structural Engineering, February 2001.
[4] Al-Tantawy, F.A. Fathelbab and M.S. Ramadan, "Finite Element Modeling of Strengthened Simple Beams using FRP Techniques: A parametric Study", International journal for the Science and Engineering of Concrete and Building materials, June 2011.
[5] A.Bashir, K. Ahmad, H.U. Khan, M.N. Rafique and S. Karam, "Identification of Shear Cracks in Reinforced Beams Using Finite Element Method", Pakistan Journal of Science, Vol. 66, No. 1, March 2014.
[6] Anthony J. Wolanski, B.S, "Flexural Behavior of Reinforced and Prestressed Concrete Beams Using Finite Element Analysis", A Thesis submitted to the Faculty of the Graduate School, Marquette University, May, 2004
[7] A.Khennane, L. Dahmani and S. Kac, "Crack Identification in Reinforced Concrete Beams Using Ansys Software", Strength of Materials, Vol. 42, No. 2, 2010
[8] I.Saifullah, M.A. Hossain, S.M.K.Uddin, M.R.A. Khan and M.A. Amin, "Nonlinear Analysis of RC Beam for Different Shear Reinforcement Patterns by Finite Element Analysis", International Journal of Civil & Environmental Engineering, Vol:11 No: 01, February 2011.
[9] Damian Kachlakev and Thomas Miller, "Finite Element Modeling of Reinforced Concrete Structures Strengthened with FRP Laminates", for Oregon Department of Transportation Research Salem, and Federal Highway Administration Washington, May 2001.