

Crack Identification and Analysis of Rcc Beam Using Ansys

M.Tech. Scholar Shubham Tiwari, Professor Dr. Rakesh Patel

Department of Civil Engineering,
SIRTS, Bhopal, MP, India

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 analyse these structural components. In this article, the reinforced concrete beam has been modelled and analysed when subjected to two point loads at one third span from each support, using FEA packages. The modelled and analysed beam having size 5000 mm × 300 mm × 450 mm with 3 numbers of 12 mm diameter bars as main reinforcement, 2 numbers of 8 mm diameter as hanger bars and 8 mm diameter at 100 mm c/c as shear reinforcement. The results of the beam with respect to mesh density, varying depths, reinforcements, crack pattern at various load conditions such as 100 KN, 250KN, 350KN, 450KN, 550KN and 650KN are analysed and discussed. Finite element software ANSYS 15.0 is used for modelling and analysis by conducting linear static analysis.

Keywords – Material nonlinearity, Finite element analysis, Convergence, Varying depths, ANSYS.

I. 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 behaviour of the structure. But it is time-consuming and expensive. Finite element analysis is also used to analyse 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 behaviour 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 behaviour of reinforced concrete beams using Finite Element modelling. 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 responds well to non-linear analysis as each component possesses different stress-strain behaviour. 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. Non-linear analysis is a method that stimulates the exact behaviour of the material to evaluate strength in inelastic range and to identify the potential of high load carrying capacity of the components through redistribution, tensile and shear strength. Nonlinear behaviour of reinforced concrete beams is complex due to various parameters.

Non-linearity may be geometric or material non-linearity. A structure can have either of the one or both of them. Material non-linearity contains non-linear stress strain relationship of material and hence modulus of elasticity is not a unique value. The geometry of the body is changed during loading in slender members such as columns and also in deformable bodies. Such case, geometric nonlinearity is encountered.

In this study, nonlinear finite element analysis is carried out using ANSYS which employs Newton Raphson method to solve higher order differential equations. Many attempts have been made by the past researchers to predict the behaviour using ANSYS. The accuracy and convergence of the solution depends on factors such as mesh density, constitutive properties of concrete, convergence criteria and tolerance values etc. Thus in the present study an attempt is made to perform nonlinear finite element analysis to analyse the reinforced concrete beam.

II. LITERATURE REVIEW

Robert R. S. and Prince A. G. (2010) studied finite element modelling on behaviour of reinforced concrete beam-column joints retrofitted with carbon fibre reinforced polymer. The Finite element modelling (FEM) has turned to be recreating the physical conduct of complex building frameworks. The (FEA) programs have increased normal acknowledgment among architects in industry and analysts. The examination of retrofitted with carbon fibre reinforced polymer sheets (CFRP) utilizing ANSYS have been exhibited in this paper. Three different strengthened sheet of CPRF on solid shaft were displayed utilizing ANSYS. Both the ends of the beam in investigation have been kept pivoted. Static load was connected at the free end of the cantilever bar. The analyses have been conducted for the retrofitted beam and the outcomes have been exhibited.

Ismail Saifullah et. al. (2011) focuses on the behaviour of reinforced concrete beam for different pattern of shear reinforcement to evaluate the effective shear reinforcement pattern and also compare the variation in behaviour of reinforced concrete beam for with and without shear reinforcement with a simulation. To carry out the analysis, six 3D beams without and with different patterns of shear reinforcement is built using comprehensive computer software ANSYS 10, 2005 SAS IP, Inc package. The static non-linear analysis is done to find out ultimate capacity, formation of first crack and its distance from support, initiation of diagonal crack and its distance from support. Load deflection response was also closely observed and compared with the result from theoretical calculation. From close observation of analyses results it was found that all types of web reinforcements were almost same effective for static loading condition.

Banu D. et al (2012) studied the numerical modelling of two-way reinforced concrete slabs strengthen with carbon fibres reinforced polymer strips. They applied FRP as an external layer to the RC (Reinforced Concrete) beams. They have used ANSYS software to analysis the effect of FRP material as an external layer to see the effect of it on load carrying capacity. They have used SOLI65 element to model the 3D concrete beams while SOLID45 has been used to design the thick shells. They have conducted their results for load-deflection and ultimate carrying capacity.

Parandaman P. and Jayaram M. (2014) studied the Finite element analysis of reinforced concrete beam retrofitted with different fibre composites. They have use Pro-E software for modelling and ANSYS for analysis the modelled geometry. The used carbon fibre reinforced polymer (CFRP) sheet for first layer of RC beam, glass fibre reinforced polymer (GFRP) sheet for second layer

and Kevlar fibre reinforced polymer (KFRP) sheet for third layer. SOLID65 element for concrete beam and SOLID45 element for FRP has been used by them. They found that deflection has been minimized around 75% compared to the conventional RC beam when CFRP used. Deflection minimized around 65% when GFRP used and 60% when KFRP used. Load carrying capacity increases by using FRP laminates and strength increases. Khushaboo and Imran Alam (2017) determined the modulus of elasticity of Reinforced Concrete by non-linear Finite Element Analysis (model size 300 x 500 x 6000 mm) with different grades of concrete (M20, M25, M30) with change percent of compression reinforced using ANSYS.

III. OBJECTIVE

The main objectives of this study are:

1. To perform non-linear analysis of RCC beam with varying load 100 KN, 250KN and 350KN.
2. To study the behaviour of RCC beam w.r.t mesh density and varying depths is analysed using ANSYS 15.
3. To analyse cracks in RCC beam using ANSYS 15.

IV. METHODOLOGY

1. Geometry of Beam

The modelled and analysed beam having size 5000 mm × 300 mm × 350 mm with 3 numbers of 12 mm diameter bars as main reinforcement, 2 numbers of 8 mm diameter as hanger bars and 8 mm diameter at 100 mm c/c as shear reinforcement has been consider for this study. The beam is simply supported with hinge at one end and roller at other end. Two point loads are applied at one-third of the span. The details of the beam are shown in Fig.1

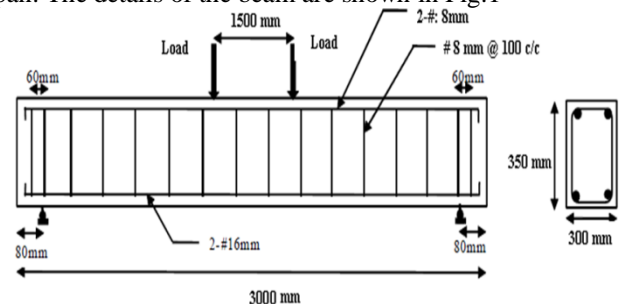


Fig. 1. Geometry of beam

2. Material Properties

The grade of reinforcement bar used beam model was Fe415, Elastic modulus was 2×105 MPa and Poison ratio -0.3, whereas concrete grade was M30, Elastic modulus was 25000 MPa and Poison's ratio was 0.15.

3. Element Used For Modeling

Concrete beam is modelled using eight node Solid65 element which has three degrees of freedom at each node.

For modelling steel reinforcement, Link180 spar element with three degrees of freedom at each node is used. Supports and loading points are modelled using eight noded Solid180 elements.

4. Modelling

The ANSYS version 15 has been used to model the beam and the analysis part also has been conducted in ANSYS.

5. Meshing

In order to obtain the actual results from the Solid 65 element, a mesh was recommended. The meshing of the reinforcement was a special case compared to volumes. The beam was meshed such that it is considered of square element of size 10mm. The necessary mesh attributes were set before the volume was meshed.

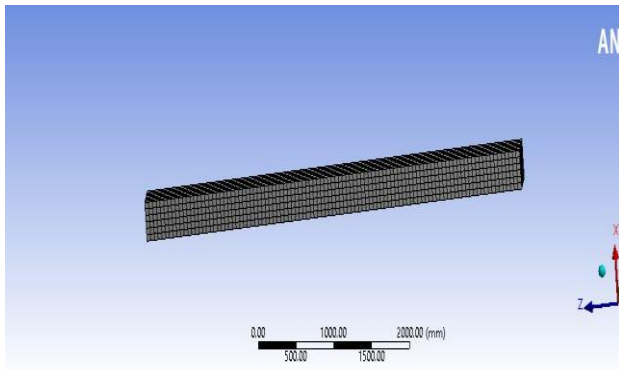


Fig. 2 Mesh model.

6. Loads And Boundary Conditions

Displacement conditions were needed to constraint the model to get a unique solution. The support was modelled as a fixed support at one end and hinged support at the other end. The external loads were applied as concentrated forces at equal one-third distance of the beam.

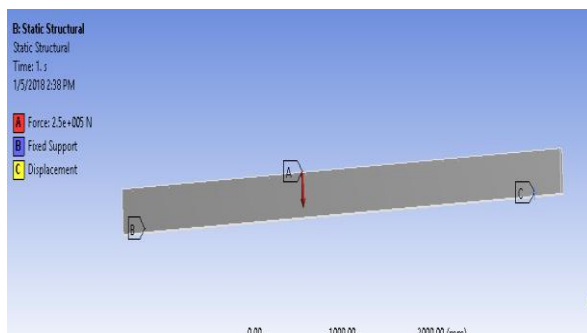
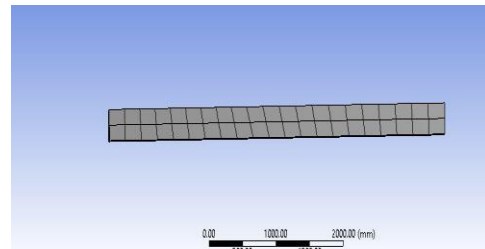


Fig. 3 Loads and boundary condition.

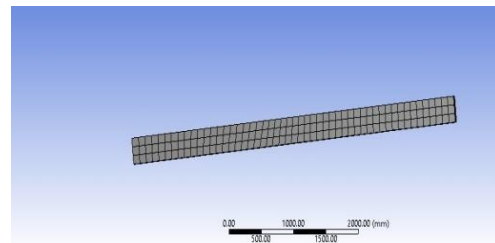
7. Convergence Study

Convergence study is performed using plain concrete beams in a non-linear analysis. A FEM model of the full plain concrete beam is considered for the study without

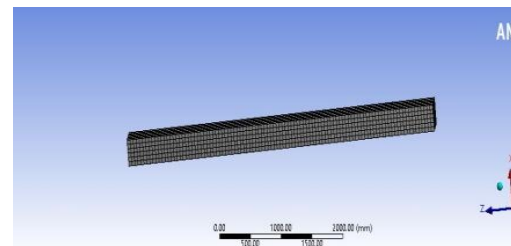
steel reinforcement to determine the appropriate mesh density. Three plain concrete beams of same size and same material properties are modelled in ANSYS 15 with increasing number of elements and number of nodes using Solid65 concrete elements.



1060 elements

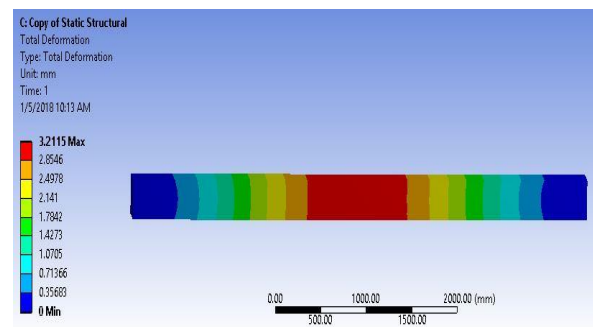


1588 elements

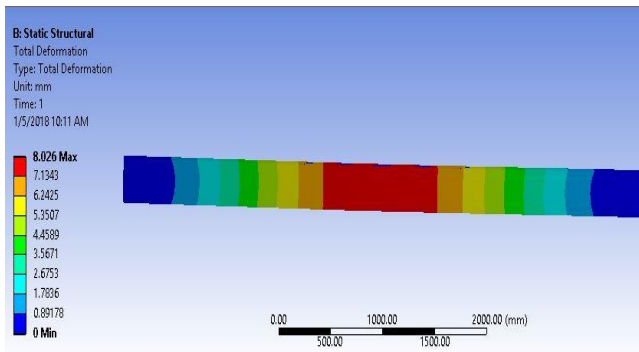


5218 elements

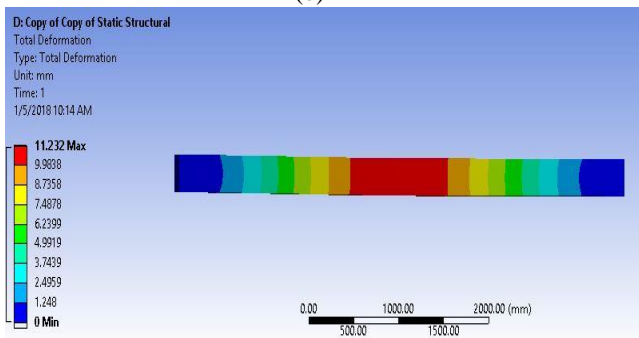
Fig. 4 Beam models used for convergence Study.



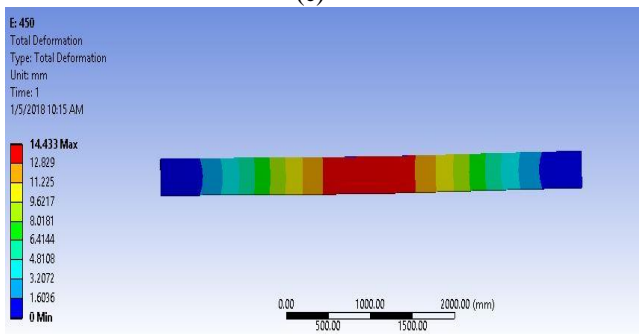
(a)



(b)



(c)



(d)

Fig. 5 Deformation diagram of the convergence study when.

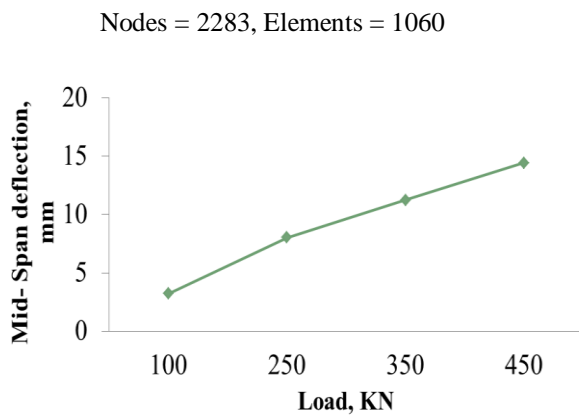
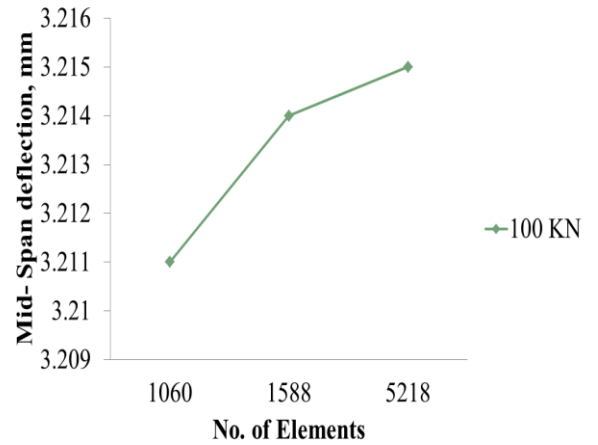
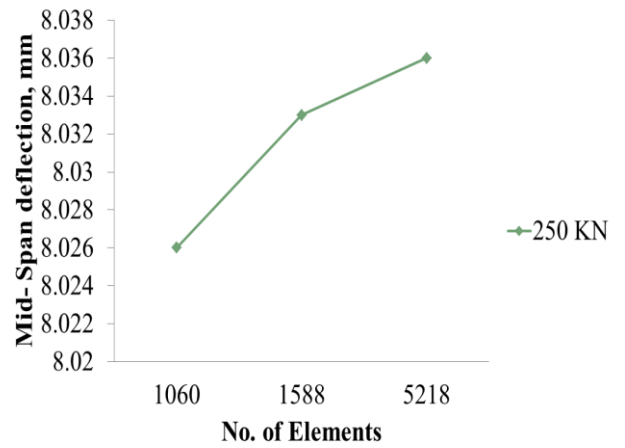


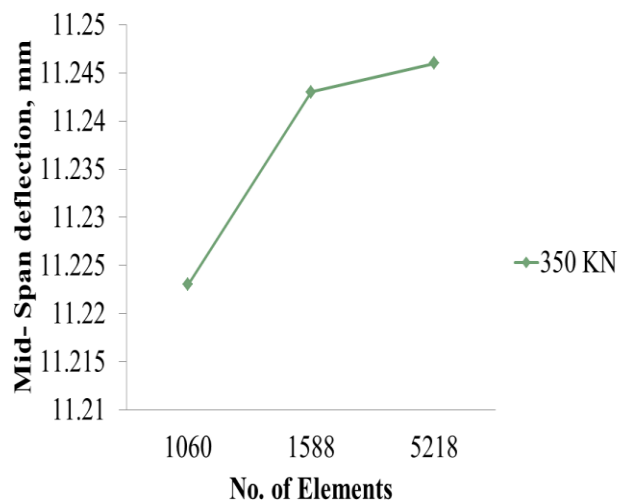
Fig. 6: Mid- Span deflection at various loads when, Nodes = 2283, Elements = 1060



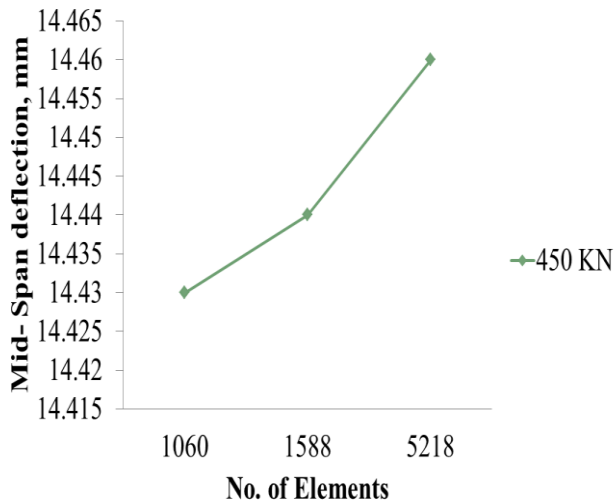
(a)



(b)



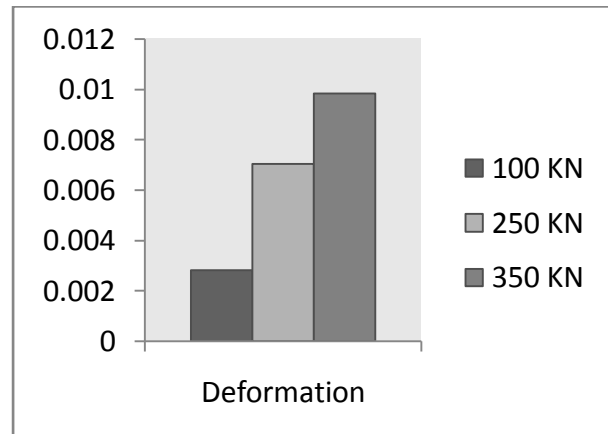
(c)



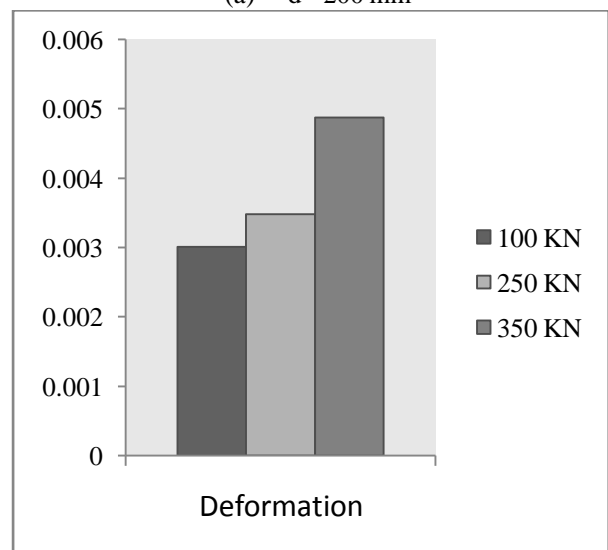
(d)
 Fig. 7: Mid- Span deflection vs no. of elements at different load

V. COMPARISON OF THE LOAD-DEFLECTION CURVE FOR DIFFERENT DEPTHS

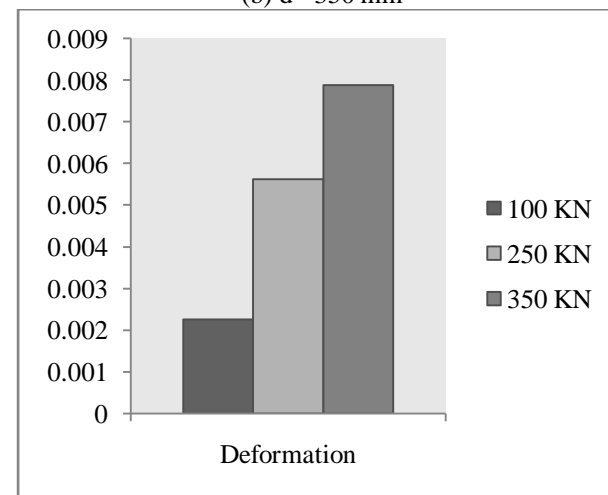
A parametric study is performed on the depth of the beam to study the behaviour of the beam. Load deflection curve for different depths of beam are done and the load at first crack is obtained. The depths adopted are 200mm, 300mm and 400mm.



(a) d= 200 mm



(b) d= 350 mm



(c) d= 400 mm

Fig. 10: Deformation in beam

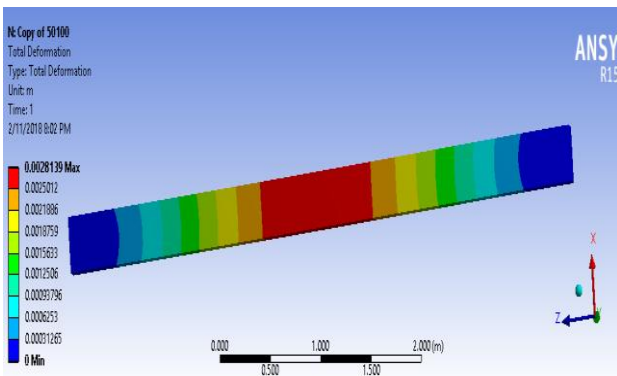


Fig. 8: Deformation diagram when d = 200mm, Force = 100 KN

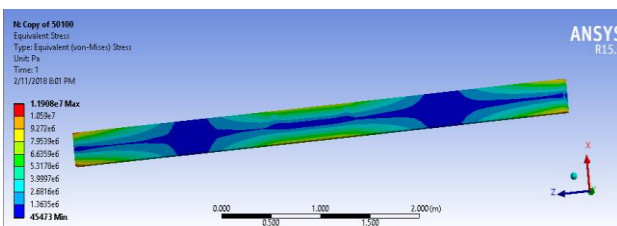


Fig. 9: Stress diagram when d = 200mm, Force = 100 KN

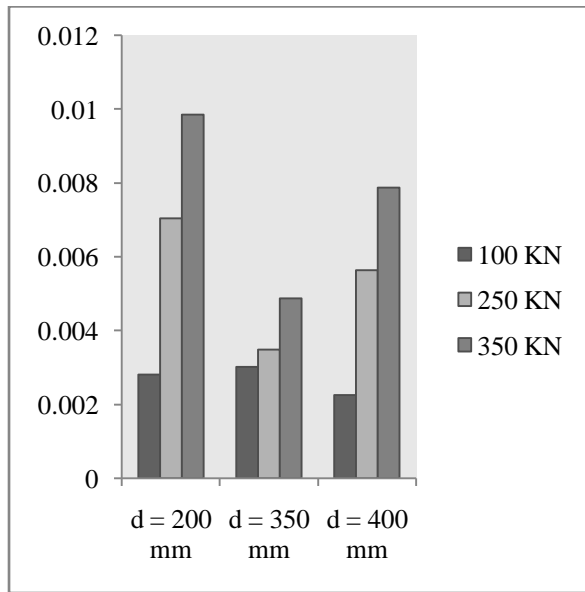


Fig. 11: Deformation in beam with different depth for different load.

Comparison of Load-Deflection curves for all the depths are shown in fig 11 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 first increases and then decreases.

VI. CRACK IDENTIFICATION IN REINFORCED CONCRETE BEAMS

For 350 mm depth, though the load at first crack is 60 KN, the deflection is more. Also for beam of 200 mm depth, load at first crack is early at 60 KN. For beam of 400 mm depth, load at first crack is about 58 KN which is moderate but the deflection is high again.



Fig. 12 First crack of the concrete model

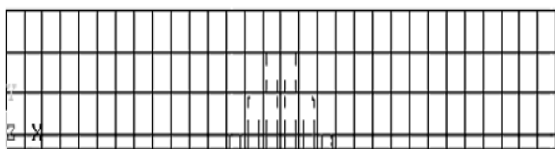


Fig. 13 Diagonal crack of the concrete model

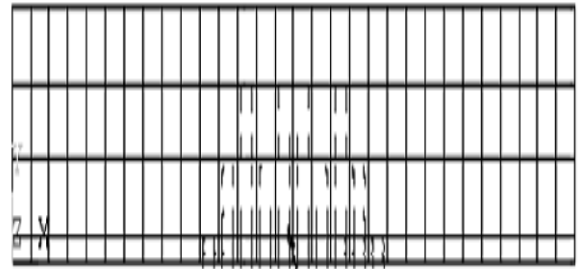


Fig. 14 Cracking at 60,000 and 70,000 N



Fig. 15 Further cracking at 80,000 and 90,000 N

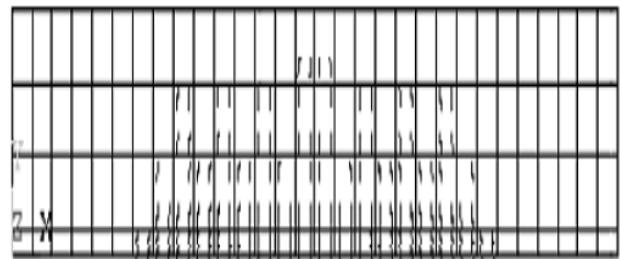


Fig. 16 Increased cracking after yielding of reinforcement

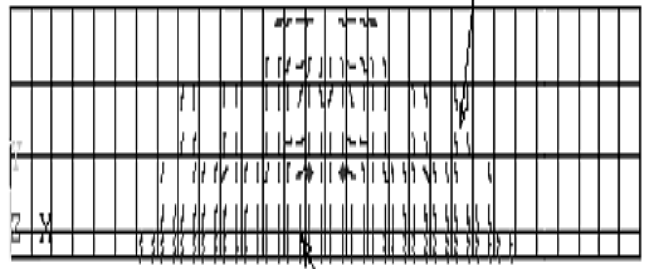


Fig. 17 Failure of the concrete beam

VII. CONCLUSIONS

In this study, the behaviour of reinforced concrete beam is analysed using finite element method. A control beam is analysed 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 and shear reinforcement. After compiling and analysing the results from each test, the following conclusions can be made:

1. Reinforced concrete beam can be modelled and analysed using ANSYS 15.0 software and obtain accurate results.
2. It is observed that the deflection is increasing from 1060 elements to 5218 elements. The observed deflection is at various load is noted. So the finite element model consisting of 5218 number of Solid 65 concrete elements is used for this entire study.
3. 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 first increases and then decreases.
4. For 350 mm depth, though the load at first crack is 60 KN, the deflection is more. Also for beam of 200 mm depth, load at first crack is early at 60 KN. For beam of 400 mm depth, load at first crack is about 58 KN which is moderate but the deflection is high again.

Engineering and Scientific Research,3(1),pp-120-126.

- [9] Nimiya, R.J., Saibabu, S. and Siva, K.P. (2014), "Finite Element Analysis of Reinforced and Pre-Tensioned Concrete Beams", 4(10), pp-449-457.
- [10] N. Sundar, P. N. Raghunath and G. Dhinakaran, Flexural Behavior of RC beams with Hybrid FRP Strengthening. International Journal of Civil Engineering and Technology, 7(6), 2016, pp.427-433.
- [11] Subramanian, T. and Jayalakshmi, J. (2015), "Analytical Investigation of Bonded Glass Fibre Reinforced Polymer Sheets with Reinforced Concrete Beam Using Ansys", International Journal of Application or Innovation in Engineering & Management, 4(5),pp-105-112.

REFERENCES

- [1] Robert R. S. and Prince A. G., Finite Element Modelling on Behaviour of reinforced concrete beam-column joints retrofitted with carbon fibre reinforced polymer, International Journal of Civil and Structural Engineering, vol. 1, Issue 3, PP: 576-582.
- [2] 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.
- [3] Banu D., Barros R. C. D. and Taranu N., Numerical Modelling of Two-Way Reinforced Concrete Slabs Strengthened With Carbon Fibber Reinforced Polymers Strips, International Conference on Experimental Mechanics, 22-27. 2012.
- [4] Parandaman P. and Jayaram M., Finite element analysis of reinforced concrete beam retrofitted with different fibre composites, Middle-East Journal of Scientific Research 22(7), 948-953, 2014.
- [5] Musmar M. A., Rjoub M. I. and Hadi M. A. A., Nonlinear Finite Element Analysis of Shallow Reinforced Concrete Beams Using Solid65 Element, ARPN Journal of Engineering and Applied Sciences, 9(2), 85- 89, 2014.
- [6] R. U. and Kulkarni D. B. Flexural Behavioural Study on RC Beam with Externally Bonded Aramid Fibre Reinforced Polymer, International Journal of Research in Engineering and Technology, 3(7), 316-321, 2014.
- [7] Jayajothi P., Kumutha R. and Vijai K., Finite Element Analysis of FRP Strengthened RC Beams Using Ansys, Asian Journal of Civil Engineering, 14(4), 631-643, 2013.
- [8] Pradeep, S. Abhishek, M. and Arpit, K. (2016), "Finite element analysis of reinforced concrete beam using Ansys", International Journal of Current