ISSN XXXX XXXX © 2018 IJESC

Research Article Volume 8 Issue No.6

Flexural Behaviour of Reinforced Concrete Beam by Finite Element Modeling

B.Venkateswarlu[1], K.Sita Hemanth[2]
PG Scholar[1], Assistant Professor[2]
Department of Civil Engineering
Chirala Engineering College, Chirala, India

Abstract:
To find out the response of individual elements of a structure, experimental based analysis has been widely used now. To understand the individual components behaviour Finite element analyses are now widely used & become the choice of modern engineering tools for the researcher. In this present study, destructive testing on simply supported beam was performed in the laboratory & load-deflection data of the reinforced concrete beams was recorded. After that finite element modelling analysis was carried out by ABAQUS ACE 6.10 by using the same material properties. Finally results from both the computer modelling and experimental data were compared. From this comparison it was found that computer based modelling is can be an excellent alternative of destructive laboratory test with an acceptable variation of results. In addition, an analytical investigation was carried out for a beam with ABAQUS ACE 6.10 with different reinforcement ratio sections (plain section, under reinforced section, balanced section and over reinforced). The observation was mainly focused on reinforced concrete beam behaviour at different points of interest which were then tabulated and compared. Maximum load carrying capacity was observed for over reinforced beam but on the other it was the balanced condition beam at ultimate load.

Keywords: Finite Element Analysis, Abaqus Ace, Destructive Test, Reinforced concrete Beam, Load and Deflection.

I. INTRODUCTION

A concrete beam is a structural element that carries load primarily in bending. Bending causes a Beam to go into compression and tension. The loads carried by a beam are transferred to columns, walls, which is then transferred to foundations. The compression section must be designed to resist buckling and crushing, while the tension section must be able to adequately resist to the tension. Experimental based testing has been widely used as a means to analyses individual elements and the effects of concrete strength under loading. While this is a method that produces real life response, it is extremely time consuming, and the use of materials can be quite costly. The use of finite element analysis to study these components has also been used. Unfortunately, early attempts to accomplish this proceeding were also very time consuming and infeasible using existing software and hardware. In recent years, however, the use of finite element analysis has increased due to progressing knowledge and capabilities of computer software and hardware. It has now become the choice method to analyses concrete structural components. The use of computer software to model these elements is much faster and extremely cost-effective. This helps in refining the analytical tools, so that even without experimental proof or check the complex nonlinear behavior of RC beams can be confidently predicted. Hence, wider attempts were made by various researchers to accurately predict the behavior of RC beams till complete failure using various FE software. It has been found that due to quasi brittle material behavior of concrete, many parameters are to be properly taken into consideration in order to obtain an accurate solution. Hence, numbers of trial analyses are carried about using ABAQUS ACE 6.10 by changing various parameters which influences the accuracy and convergence. Idealization of reinforcement in concrete, constitutive properties of concrete, mesh density, incorporation of boundary conditions for supports and symmetric planes, modeling of loading and support regions, effect of shear reinforcement on flexural behavior, effect of convergence criteria, impact of percentage of reinforcement and other parameters which governs the analysis are considered for the present study. The results and discussion of the present study are compared with the findings available in the literature. Reinforced concrete (RC) has become one of the most important building materials and is widely used in many types of engineering structures. The economy, the efficiency, the strength and the stiffness of reinforced concrete make it an attractive material for a wide range of structural applications. For its use as structural material, concrete must satisfy the following conditions:

(1) ***The structure must be strong and safe*****.** The proper application of the fundamental principles of analysis, the laws of equilibrium and the consideration of the mechanical properties of the component materials should result in a sufficient margin of safety against collapse under accidental overloads.

(2) ***The structure must be stiff and appear unblemished*****.** Care must be taken to control deflections under service loads and to limit the crack width to an acceptable level.

(3) ***The structure must be economical*****.** Materials must be used efficiently, since the difference in unit cost between concrete and steel is relatively large. The ultimate objective of design is the creation of a safe and economical structure. Advanced analytical tools can be an indispensable aid in the assessment of the safety and the serviceability of a proposed design. This is, especially, true for many complex modern structures such as nuclear power plants, bridges, off-shore platforms for oil and gas exploration and underground or underwater tunnels, which are subjected to very complex load histories. The safety and serviceability assessment of these structures necessitates the development of accurate and reliable methods and models for their analysis. In addition, the rise in cost of structures encourages engineers to seek more economical alternative designs often resorting to innovative construction methods without lowering the safety of the structure. Intimately related to the increase in scale of modern structures is the extent and impact of disaster in terms of human and economic loss in the event of structural failure. As a result, careful and detailed structural safety analysis becomes more and more necessary. The objective of such an analysis is the investigation of the behaviour of the structure under all possible loading conditions, both, monotonic and cyclic, its time-dependent behaviour, and, especially, its behaviour under overloading.

II. EXPERIMENTAL INVESTIGATION

When the maximum stresses in steel and concrete simultaneously reaches allowable value the section is called balanced section when the percentage of steel in a section is less than that required for a balanced section it is under reinforced section when the percentage of steel in a section is more than that required for a balanced section it is over reinforced section. An under-reinforced beam is one in which the tension capacity of the tensile reinforcement is smaller than the combined compression capacity of the concrete and the compression steel (under-reinforced at tensile face).When the reinforced concrete element is subject to increasing bending moment, the tension steel yields while the concrete does not reach its ultimate failure condition. As the tension steel yields and stretches, an "under-reinforced" concrete also yields in a ductile manner, exhibiting a large deformation and warning before its ultimate failure. In this case the yield stress of the steel governs the design. An over-reinforced beam is one in which the tension capacity of the tension steel is greater than the combined compression capacity of the concrete and the compression steel (over-reinforced at tensile face). So the "over-reinforced concrete" beam fails by crushing of the compressive-zone concrete and before the tension zone steel yields, which does not provide any warning before failure as the failure is instantaneous.

$$\frac{x_u}{d} = \frac{0.87 f_y A_{st}}{0.36 f_{ck} b d}$$

$\frac{x_u}{d} = \frac{x_{u\,max}}{d}$ Is called balanced section

$\frac{x_u}{d} < \frac{x_{u\,max}}{d}$ Is called Under Reinforced section

$\frac{x_u}{d} > \frac{x_{u\,max}}{d}$ Is called Over Reinforced section

III. SPECIMEN DESCRIPTION

Two plain concrete beams, Six Reinforced concrete beams of M30 grade OPC concrete (1:1.34:2.88) used in this investigation program were it 100 mm wide x 200 mm deep x 1200 mm long. Two beams (Under reinforced) are casted with 2- 12Ø bars at bottom and 2-8Ø hanger bars, 8Ø @135 stirrups are provide, And Two beams (Balanced section) are casted with 2- 12Ø, 2-8Ø bars at bottom and 2-8Ø hanger bars, 8Ø @135 stirrups are provide, Balance Two beams (Over reinforced) are casted with 2- 16Ø bars at bottom and 2-8Ø hanger bars, 8Ø @135 stirrups are provide these details are mention in Table I.

Table .1. Details of Beams Geometry

Section	Mix	W/C	F_{ck} n/mm^2	F_y n/mm^2	Ast (mm^2)	Size of the beam	Design load kn
Under	1: 1.34 :2.88	0.41	38.24	415	226.2	1200* 200* 100	70.4
Balanced	1: 1.34: 2.88	0.41	38.24	415	326.1	1200* 200* 100	91.5
Over	1: 1.34: 2.88	0.41	38.24	415	402.6	1200* 200* 100	103.5
Plane	1: 1.34: 2.88	0.41	38.24	……	……	1200* 200* 100	………

EXPERIMENTAL PROCEDURE

After completion of 28 days curing the beams were subjected to three point loading by using UTM. Dial gauges were arranged to the bottom of beam. And hydraulic load is applied linearly we measure the deflection of beam at each interval of 5KN load , and also observe the first crack at which load and corresponding deflection this process is continue for all beams and also note the ultimate load with corresponding deflection for all beam sections.

FLEXURAL TESTING OF BEAM

The experimental program consisted of testing of eight prisms of size 100x200x1200mm of M30 grade concrete subjected to three point loading which is used two prisms of plane concrete, two under reinforced concrete beams, two balanced reinforced concrete beams and two over reinforced concrete beams UTM machine where two roller supports are used as shown in Figure

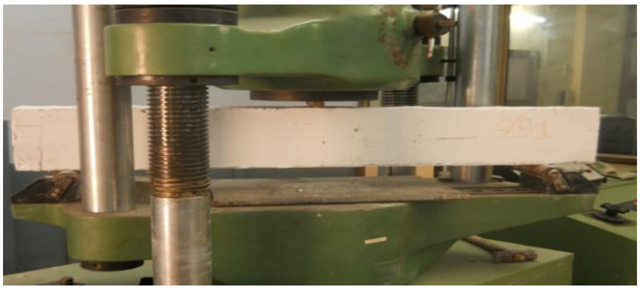

Figure.1. Three point loading test using UTM machine

Figure.2. Dial gauge arrangement for testing beam

ANALYTICAL MODEL

In most finite element software packages like ABAQUS, ADINA, ANSYS, COSMOS/M, DIANA, LS- DYNA, LUSAS and NASTRAN provide different types of elements for one- ,two- or three-dimensional problems such as plane stress, plane strain, three dimensional solid element, straight and curved beams, and shell elements. The finite element method is one of the techniques used for numerical solutions in the field of ordinary differential equations. Huebner (1984) explicate that performance of each finite element procedure can be summarized by:

- **Sub-divide the model:** The first step is to sub-divide the element into tiny elements, in this case a variety of shapes such as triangular or rectangular.
- **Select interpolation function:** The next step is to assign nodes to each element and then choose the type of interpolation function required to vary them. This vector can be a vector or higher–order of tensor.
- **Find the interpolation properties:** After creating the model, the matrix equation which covers all the material properties can be presented.
- **Assemble the element properties to obtain a system of equations:** At this stage the matrix attributed to every element will be assembled to achieve a global matrix in the system.
- **Solve the system of equations:** In this case, on the basis of method of solution (linear-algebra), the system of equation will be solved simultaneously.

GENERAL DESCRIPTION OF ABAQUS

ABAQUS is a complete, modular, self-contained finite element system. The system is capable of solving linear, nonlinear, static, and dynamic problems including fields of heat transfer, fluid, mechanic and electromagnetic problems.
Architects and developers often face the challenge of performing under severe operational constraints. Regulatory compliance is a persistent challenge. Simulation has been widely used in the construction area in order to optimize productivity and resource allocation. SIMULIA offers ABAQUS Unified FEA solutions for predicting the strength and deformations in structures in the linear and nonlinear regime. ABAQUS is a highly sophisticated, general purpose finite element program, designed primarily to model the behavior of solids and structures under externally applied loading.

ABAQUS includes the following features:

- Capabilities for both static and dynamic problems
- The ability to model very large shape changes in solids, in both two and three dimensions
- A very extensive element library, including a full set of continuum elements, beam elements, shell and plate elements, among others.
- A sophisticated capability to model contact between solids
- An advanced material library, including the usual elastic and elastic – plastic solids; models for foams, concrete, soils, piezoelectric materials, and many others.
- Capabilities to model a number of phenomena of interest, including vibrations, coupled fluid/structure interactions, acoustics, buckling problems, and so on.

SIMULIA provides realistic simulation solutions that allow accurate predictions of strength and deformations in structures in the linear as well as nonlinear regime. The available pre and post processing technology coupled with sophisticated analyses solvers provide a complete and reliable solution for structural analysis. Static, modal, and dynamic analyses can be performed using a wide range of material models and loading/boundary conditions. Buildings along with the foundations can be modeled simultaneously and detailed effects due to contact and damage can be included in the analyses.

SOLUTION CAPABILITIES

- A wide range of material models including metal plasticity and concrete damage
- Nonlinear implicit and explicit dynamic analysis capabilities
- Modal and buckling analysis, response spectrum analysis, random response analysis
- Embedded elements and rebar with pre-stress for modeling reinforcement

It delivers accurate, high performance solutions for challenging nonlinear problems, large scale linear dynamics applications & routine design simulations. ABAQUS is used in the automotive, aerospace, and industrial products industries. The product is popular with academic and research institutions due to the wide material modeling capability, and the program's ability to be customized. ABAQUS also provides a good collection of multiphase capabilities, such as coupled acoustic-structural, piezoelectric, and structural-pore capabilities, making it attractive for production-level simulations where multiple fields need to be coupled. ABAQUS was initially designed to address non-linear physical behavior; as a result, the package has an extensive range of material models such as elastomeric (rubber like) material capabilities.

NUMERICAL MODEL PREPARATION

On the basis of the central objectives of this research, three dimensional Finite Element models of reinforced concrete beam were developed, and the various items concerned with modeling is addressed as follows.

- Elements type
- Material property
- Assigning sections
- Defining step
- Interaction between elements
- Specify boundary conditions and load
- Meshing
- Assigning job
- Evaluating the results

SELECTION OF ELEMENT TYPE

The numerical simulation of a reinforced concrete structure requires an accurate model of the structural elements and its constituent members acting as a composite made up of concrete and steel. A sketch of each section is created separately with ABAQUS, which can then be extruded in any direction; this is why a 3D solid element in “modeling space” using deformable type for beam was created. In order to develop concrete beam, 8- node continuum solid element was utilized.

Figure 3 shows a three –dimensional view of the model, which was used to develop the concrete beam. According to the ABAQUS users’ manual, the eight-node continuum elements (C3D8R) are formulated based on a Lagrangian description of behavior where the element deforms with the material deformation.

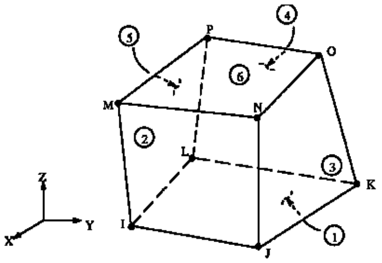

Figure.3. C3D8R-3D solid element (ABAQUS)

The solid element has eight nodes with three degrees of freedom at each node – translations in the nodal x, y, and z directions. The element is capable of plastic deformation, cracking in three orthogonal directions, and crushing. Necessary partitions of the concrete beam (of size 1200 x200 x100) are made to facilitate load application and meshing as shown in the Figure 4.

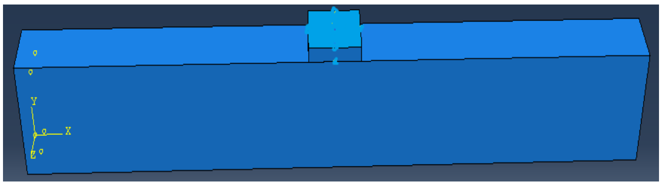

Figure.4. Developed solid model

The steel reinforcement of size 1150 mm is modeled as two – node beam elements connected to the nodes of adjacent solid elements. Each node has three degrees of freedom, – translations in the nodal x, y, and z directions. The element is also capable of plastic deformation. The geometry and node locations for this element type are shown in Figure 5.

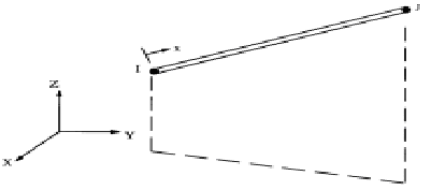

Figure .5. B31-3D Spar (ABAQUS)

On the basis of the sections available in the ABAQUS commercial package library, any profile can be assigned by regarding their shape and dimensions. The orientation of the beam cross section is defined which is available default in ABAQUS. For simplicity, stirrups, which are used to ensure beams do not fail in shear during experiment, were not considered in the simulations.

ASSEMBLY

An assembly is a collection of positioned part instances. A part instance is a usage of a part within the assembly.

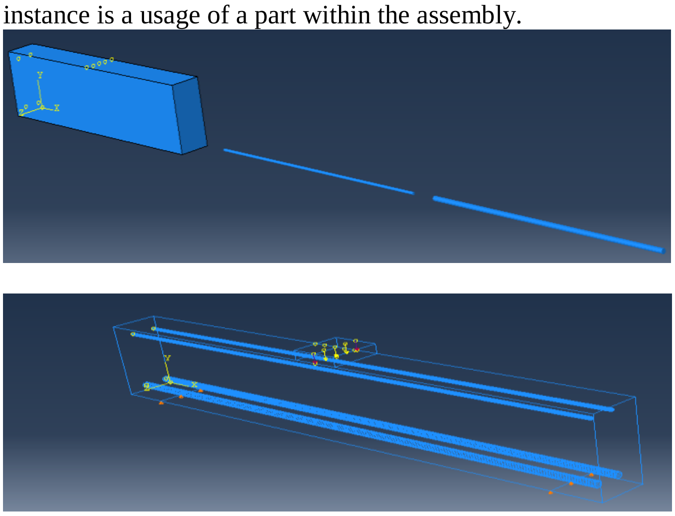

Figure.6. Assembling of part instances

All characteristics (such as mesh and section definitions) defined for a part become characteristics for each instance of that part—they are inherited by the part instances. Each part instance is positioned independently within the assembly. Linear pattern, translation and rotate tools are used to place the steel reinforcement, their respective position as in experimental beam. After assembling and assigning the properties, an input file is created which is then imported to create an orphan mesh. An orphan mesh contains nodes and elements but no geometry. This is useful for creating surfaces on concrete to apply load and also for applying boundary condition on nodes. Figure 6 shows assemblage of all parts instances.

MESHING

Meshing is the process of generating nodes and elements. A mesh is generated by defining nodes and connecting them to define the elements. In ABAQUS the capabilities of meshing in any model is classified into three main categories, the function for assigning mesh attribution, the function for mesh generation, and the function for mesh verification. There are also a number of techniques to subdivide any type of elements into tiny mesh elements; however the structural meshing is the most flexible. The first step is to determine the number of seeds per edge, and by using this option the feasibility of distributing seeds uniformly along an edge can be seen. Moreover For instance, Table II presents the quality and quantity of seeds per edge in a Reinforced Concrete beam.

Table.2. Details of mesh

Part	Mesh size	No. of nodes	No. of elements	Element type
Concrete beam	25	2135	1440	Linear hexahedral elements of type C3D8R
Steel	5	231	230	Linear line elements of type B31
Steel plate	25	50	16	Linear hexahedral elements of type C3D8R

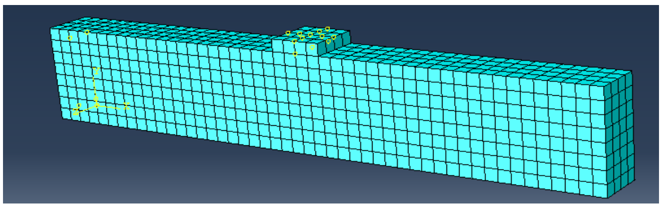

Figure .7. Meshing of concrete beam

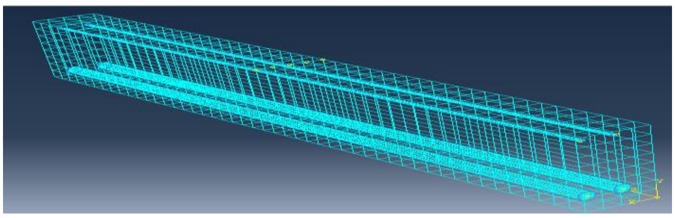

Figure .8. Meshing of steel bars

Among linear- order elements there are two types that have been considered in the analysis, i.e. fully and reduced integrated elements. In problems when a structure is subjected to bending fully integrated linear elements give poor results due to shear locking. To diminish the problem with shear locking in linear solid elements reduced integration is used. This method consists of using one fewer integration point in each direction, i.e. linear solid element has only one integration point. The adhesive is meshed using the same element but with fine mesh than concrete beam.

IV. RESULTS AND DISCUSSION

Two plane concrete beams, six reinforced concrete beams were tested to study the maximum deflections, stress in concrete, steel and crack propagations under flexure loading. The plane concrete beams are failed at minimum loads (10.31 KN) because a bottom fiber of plane concrete beam reaches maximum tensile stress and plane concrete beam have fully brittle nature. Next test the reinforced concrete beams those six beams are separated three groups according to percentage of reinforcement provide (balanced, under, over reinforced beams). The test was conducted until the beam fails and deflections were measured up to failure load. The under reinforced concrete beams are tested by three point loading system, first crack was found at 30 KN load and ultimate load carrying capacity of section is 73.33 KN for balanced section first crack found at 34 KN, ultimate load carrying capacity of section is 88.8KN, and over reinforced section first crack found at 36 KN, ultimate load carrying capacity of 86 KN From experimental testing over reinforced concrete beam section have more flexural cracks and its failed with less warning's compare to other sections and balanced section gives greater warnings its means steel is yielding in under reinforced section. From analytical investigation we observed first concrete is reaches maximum tensile stress at first crack after first crack in beam maximum load is taken by reinforcement and under reinforced section steel reaches ultimate tensile stress but over reinforced beam section steel not reach ultimate tensile stress because concrete is failed.

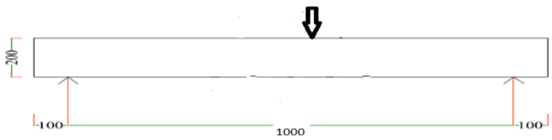

Figure.9. Testing of Plane/Reinforced concrete beam by three point loading

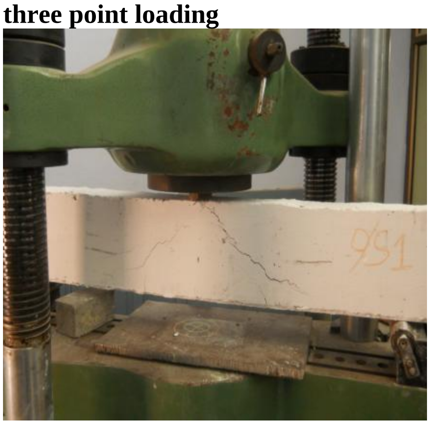

Figure.10. Flexure Failure of under reinforced concrete beam

Figure.11. Flexure Failure of over reinforced concrete beam

Figure.12. Flexure Failure of over/balanced/under reinforced concrete beam

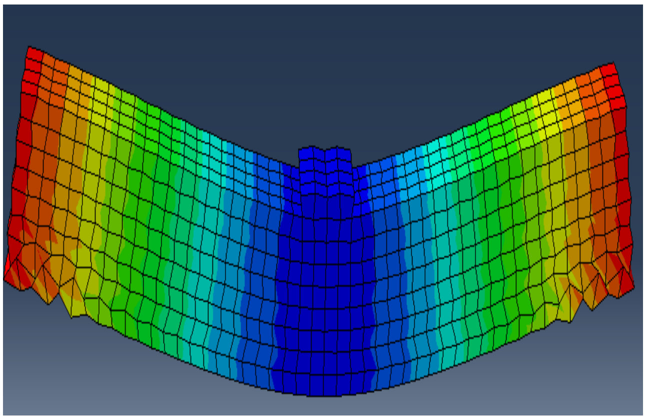

Figure.13. Deformation of Plane Concrete Beam

Table.3. Results of Plane/Over/Balanced/Under Reinforced Concrete Beam

Name of	Deflection		stress in concrete		stress in steel		1st	Ultimate load	
Section	FEA	Experimental	Tension	compression	Tension	crack	Crack at kN	FEA	Experimental
Under	5.2	5.5	3.45	12.25	427.52	172.4	30	75.1	73.3
Balanced	6.1	5.75	3.4	11.54	402.54	142.4	34	78.4	88.8
Over	5.8	6.5	3.54	11.32	362.45	121.3	36	80.4	86.2
Plane	0.4	0.45	3.32	6.6	0	0	11	11.2	10.31

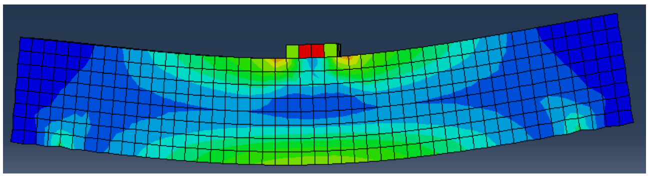

Figure.14. Stress in Plane Concrete Beam

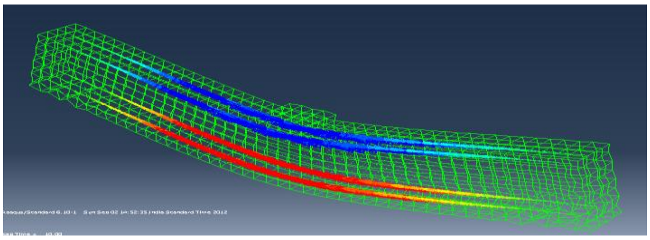

Figure.15. Stress in Steel Bar in Under Reinforced Concrete Beam

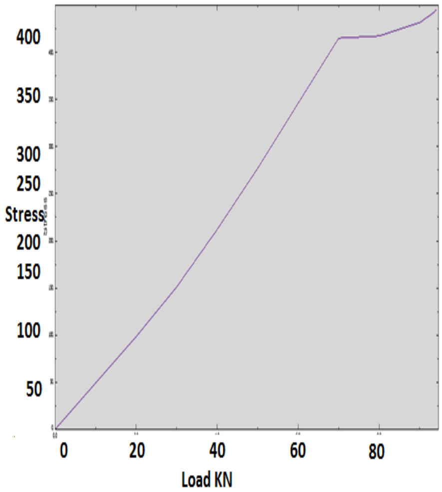

Figure.16. Stress in steel node

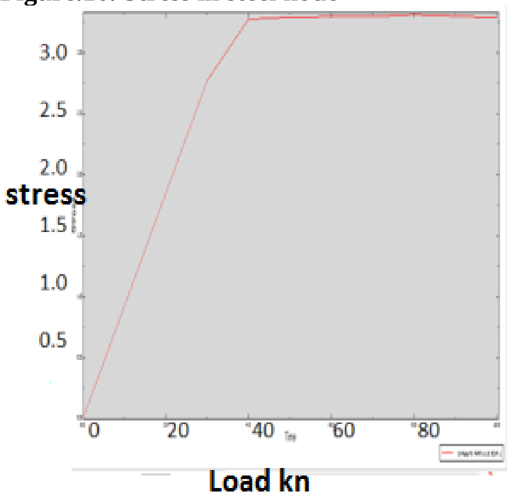

Figure.17. Stress in concrete node

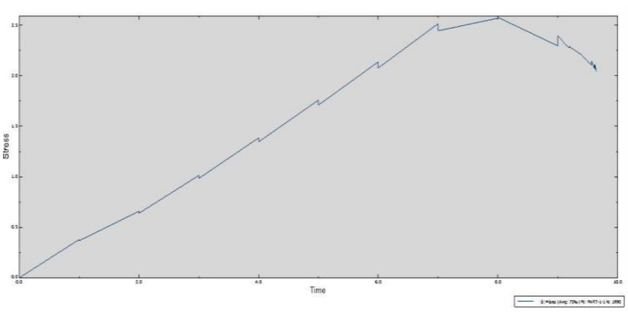

Figure.18. Stresses in Plane Concrete Beam Section

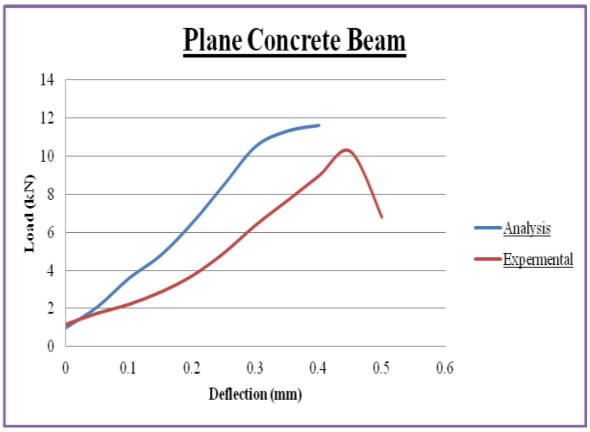

Figure.19. Load vs Deflection for Plane Section

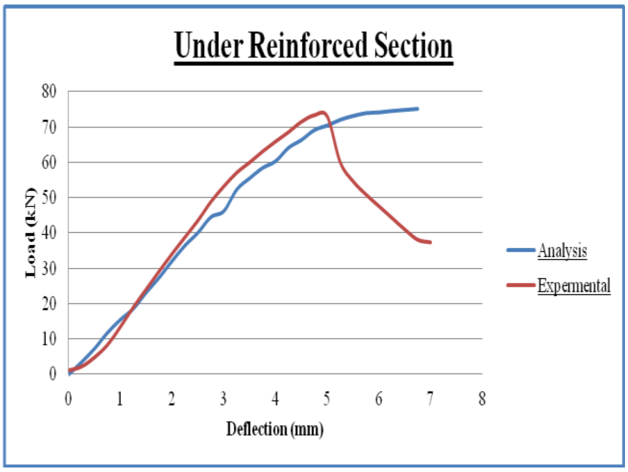

Figure.20. Load vs Deflection for Under Reinforced Section

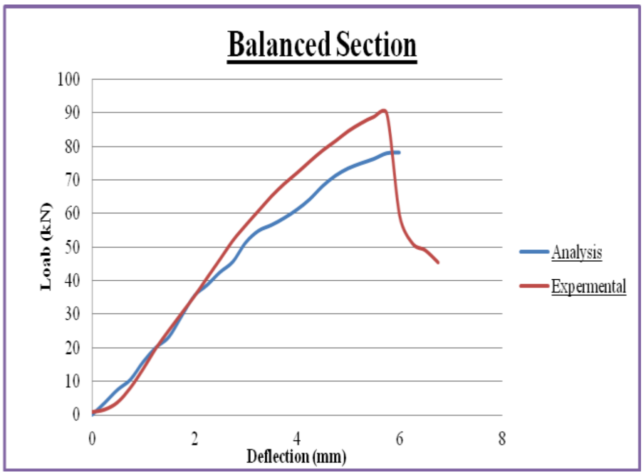

Figure.21. Load vs Deflection for Balanced Section

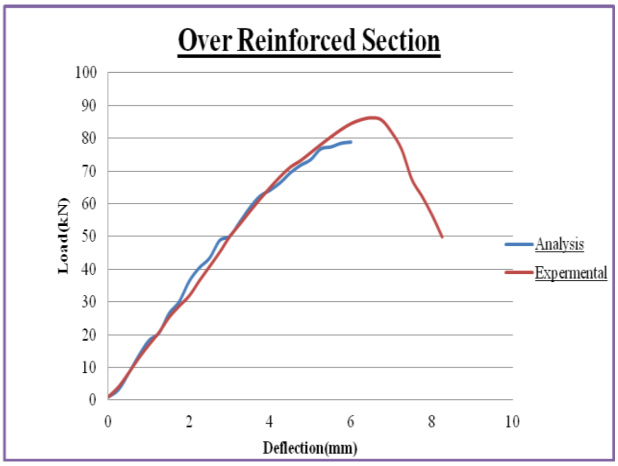

Figure.22. Load vs Deflection for Over Reinforced Section

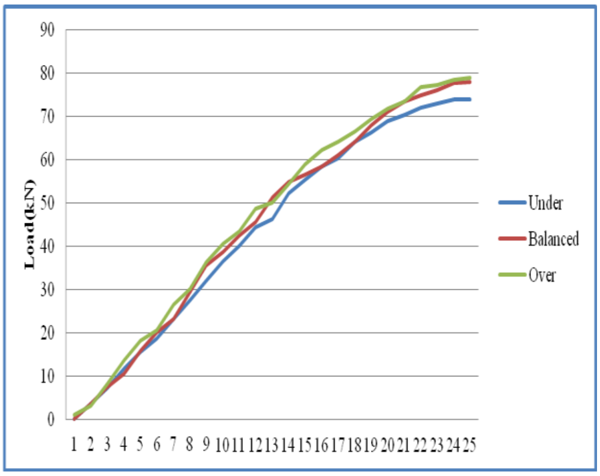

Figure.23.Comparison of Over/Balanced/Under Reinforced Beams from ABAQUS

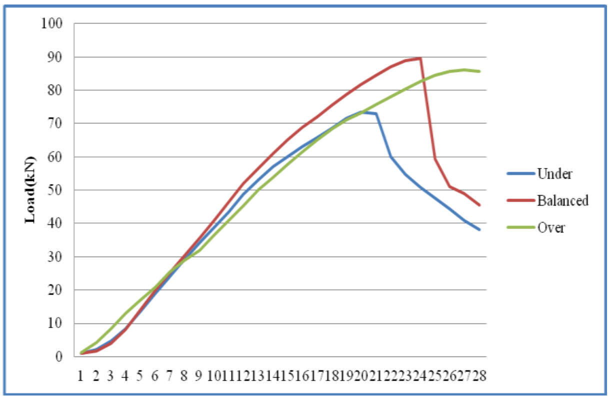

Figure.24.Comparison of Over/Balanced/Under Reinforced Beams from Experimental

V. CONCLUSION

The following conclusions can be stated based on the evaluation of the analyses of the calibration model.

1. Deflections and stresses at the centerline along with initial and progressive cracking of the finite element model compare well to experimental data obtained from a reinforced concrete beam.
2. The ultimate load carrying capacity of Plane concrete beam is 0.14 times under reinforced beam
3. The failure mechanism of a reinforced concrete beam is modeled quite well using FEA and the failure load predicted is very close to the failure load measured during experimental testing.
4. In under reinforced beam maximum elements reach ultimate stress compare to over reinforced concrete beam
5. From the analytical investigation it was observed that under reinforced ratio is the best type of reinforcement ratio among the others since it shows greatest warning zone before failure.
6. From the analytical investigation, it was observed that under reinforced section reinforcement reaches ultimate stress (415 n/mm^2), and over reinforced section reach 87% of ultimate stress.

VI. REFERENCES

[1].R. Srinivasan and K. Sathiya flexural Behavior of reinforced concrete beam using finite element analysis Concrete structural components exist in buildings.

[2].Anthony J. Wolanski, B.S. (2004) “Flexural Behavior of Reinforced and Pre-stressed Concrete Beams Using Finite Element Analysis”, Master’s Thesis, Marquette University Milwaukee, Wisconsin.

[3].Saifullah M.A. Hossain Experimental and Analytical Investigation of Flexural Behavior of Reinforced Concrete Beam

[4].Vasudevan. G 1 , Kothandaraman.S 2 Parametric study on Nonlinear Finite Element Analysis on flexural behaviour of RC beams using ANSYS

[5].Mohammed Sh. Mahmood Finite Element Modeling of Reinforced Concrete Beams Strengthened with FRP Laminates

[6].Hsuan-Teh Hu *, Fu-Ming Lin, Yih-Yuan Nonlinear finite element analysis of reinforced concrete beams strengthened by fiber-reinforced plastics

[7].Hamid Sinaei*, Mahdi Shariati, Amir Hosein Abna, Mohammad Aghaei and Ali Shariati Evaluation of reinforced concrete beam behaviour using finite element analysis by ABAQUS

[8].R. Balamuralikrishnan1,* andC. Antony Jeyasehar Flexural Behavior of RC Beams Strengthened with Carbon Fiber Reinforced Polymer (CFRP) Fabrics

[9].Nitereka, C., and Neale, K.W. (1999). Analysis of Reinforced Concrete Beams Strengthened in Flexure with Composite Laminates. Canadian Journal of Civil Engineering, 646-54.

[10].Kachlakev, D., PhD., Miller, T, PhD, P Yim, S, PhD, PE; Chansawat, K. and Potisuk, T., (2001) “Finite Element Modeling of Reinforced Concrete Structures Strengthened with FRP Laminates”, Oregon Department of Transportation, Research Group.

[11].Kasidit Chansawat.et.al., FE Models of GFRP and CFRP Strengthening of Reinforced Concrete Beams. Submitted to Advances in Civil Engineering.

[12].Nilson, A. H., Darwin, D., and Dolan C. W.,Edition, (2006), “Design of Concrete Structure”, McGraw-Hill Education(Asia),Singapore.

[13].Vazirani, V.N.; Ratwani, M.M., (1995) “Concrete Structures” , Khanna Publishers, Delhi.

Murdock, L. J., Brook, K. M. and Dewar, J. D., “Concrete: Materials and Practice”, 6th Edition, Edward Arnold, London, 1991

[14].American Concrete Institute, “Material and General Properties of Concrete,” ACI Manual of Concrete Practice, Part 1, 1996

[15].Altenbach.H, Altenbach.J & Kissing.W, (2004). Mechanics of Composite Structural Elements. Ref No: ISBN 3-540-40865-7

[16].Nakasone, Y.; Yoshimoto, S.; Stolarski, T. A., 2006, “ENGINEERING ANALYSIS WITH ANSYS SOFTWARE”, ELSEVIER, 1st Published.

[17].Hossain. Nadim, M, 1998, “Structural Concrete Theory & Design”, Addison-Wesley Publishing Company

[18].Taljsten, B. (1997). Strengthening of Beams by Plate Bonding. Journal of Materials in Civil Engineering, 9, 4, 206-12.

[19].Teng, J.G., Zhang, J.W., and Smith, S.T. (2002). Interfacial Stresses in Reinforced Concrete Beams Bonded with a Soffit Plate: A Finite Element Study. Construction and BuildingMaterials, 16, 1-14.

[20].Yang, Z.J., Chen, J.F., and Proverbs, D. (2003). Finite Element Modeling of Concrete Cover Separation Failure in FRP Plated RC Beams. Construction and Building Materials,

[21].Qing-Sheng Yang.et.al.(2002) Analytical and numerical investigation of interfacial stresses of FRP–concrete hybrid structure. Composite Structures 57 (2002) 221–226

[22].Nitereka, C., and Neale, K.W. (1999). Analysis of Reinforced Concrete Beams Strengthened in Flexure with Composite Laminates. Canadian Journal of Civil Engineering, 646-54.

[23].Vecchio, F.J., and Bucci, F. (1999). Analysis of Repaired Reinforced Concrete Structures. Journal of Structural Engineering, 125, 6, 644-52.

[24].Kasidit Chansawat.et.al., FE Models of GFRP and CFRP Strengthening of Reinforced Concrete Beams. Submitted to Advances in Civil Engineering.