EXPERIMENTAL ANALYTICAL AND INVESTIGATION OF FLEXURAL BEHAVIOR OF REINFORCED CONCRETE BEAM

PEDDI HEMA ARPITHA CHOWDARY[1], SREENU BHUVANAGIRI[2]
[1]Department of CIVIL Engineering, JNTU KAKINADA, ONGOLE, INDIA
PG SHOLAR[1] hemaarpitha111@gmail.com[1]
ASST PROF CIVIL2, JNTU KAKINADA, ONGOLE, INDIA
SREENUBHUVANAGIRI@gmail.com

ABSTRACT:Experimental based analysis has been widely used as a means to find out the response of individual elements of structure. To study these components finite element analyses are now widely used & become the choice of modern engineering tools for the researcher. In the present study, destructive test on simply supported beam was performed in the laboratory & load-deflection data of that under-reinforced concrete beams was recorded. After that finite element 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 (plain, under, balanced, over). 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:**Concrete beam, ABAQUS ACE, destructive test, the laboratory & load-deflection data, FEM***

1. INTRODUCTION

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 behaviour of RC beams can be confidently predicted. Hence, wider attempts were made by various researchers to accurately predict the behaviour of RC beams till complete failure using various FE software. It has been found that due to quasi brittle material behaviour 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, modelling of loading and support regions, effect of shear reinforcement on flexural behaviour, 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.

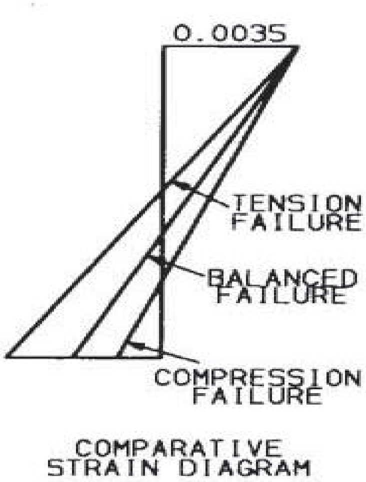
(2) *The structure must be stiff and appear unblemished*. Care must be taken to control deflections under service loads and to limit the crack width to an acceptable level.(3) *The structure must be economical*. Materials must be used efficiently, since the difference in unit cost between concrete and steel is relatively large. The ultimate objective of design is the creation of a safe and economical structure. Reinforced concrete structures are commonly designed to satisfy criteria of serviceability and safety. In order to ensure the serviceability requirement it is necessary to predict the cracking and the deflections of RC structures under service loads. In order to assess the margin of safety of RC structures against failure an accurate estimation of the ultimate load is essential and the prediction of the load-deformation behaviour of the structure throughout the range of elastic and inelastic response is desirable. Within the framework of developing advanced design and analysis methods for modern structures the need for experimental research continues. Experiments provide a firm basis for design equations, which are invaluable in the preliminary design stages. Experimental research also supplies the basic information for finite element models, such as material properties. In addition, the results of finite element models have to be evaluated by comparing them with experiments of full-scale models of structural sub assemblages or, even, entire structures. The development of reliable analytical models can, however, reduce the number of required test specimens for the solution of a given problem, recognizing that tests are time-consuming and costly and often do not simulate exactly the loading and support conditions of the actual structure.The development of analytical models of the response of RC structures is complicated by the following factors:

• Reinforced concrete is a composite material made up of concrete and steel, two materials with very different physical and mechanical behaviour;
• Concrete exhibits nonlinear behaviour even under low level loading due to nonlinear material behaviour, environmental effects, cracking, biaxial stiffening and strain softening;
• Reinforcing steel and concrete interact in a complex way through bond-slip and aggregate interlock.

2.1. Experimental Program

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.

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



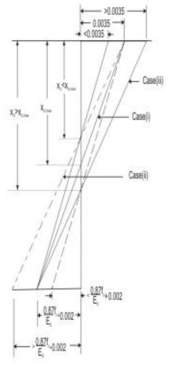


Fig.3.5.1: Strain of steel for three cases $x_u <=> x_{u,max}$

$\frac{x_u}{d} = \frac{x_{u\,max}}{d}$ 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

2.2 Description OF Specimens

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 3.1

Table 3.1 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	…….

Fig 3.1 various types of beams

2.3Flexural 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

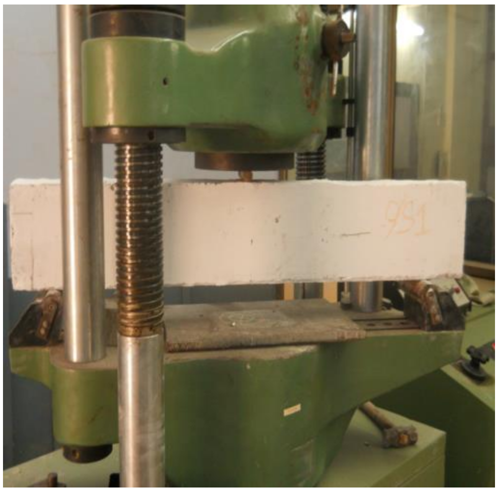


Fig.3.2 Three point loading test using UTM machin

Fig.3.3 Dial gauge arrangement for testing beam

2.4 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.

4.2 ADVANTAGES AND DISADVANTAGES OF FINITE ELEMENT TOOL

Advantages

- Includes a large library of finite elements to enable efficient and detailed modeling of virtually all modes of structural behavior.
- Encloses facilities to allow users to write user defined subroutines.
- Enables modeling of many of the special features of structural behavior of buildings.
- Provides technical support and adequate documents about using the program.

Disadvantages

- Require substantial initial investment for buying the software and associated hardware, and training up a specialist analyst.
- Costly maintenance for renewing the license and retaining the specialist analyst.

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.

ABAQUS has been chosen for the purpose of modelling and analyzing the concrete beam with steel in this study due to its flexibility in creating geometry and material modelling. This chapter describes step by step of the modelling procedures, includes all the parameters in the analysis, from the geometrical modelling until the determination of the strain from the interface. ABAQUS (Version 6.10) has been chosen for the purpose of modelling and analyzing the concrete beam steel in this study due to its flexibility in geometry and material modelling. The chapter describes step by step of the modelling procedures, includes all the parameters in the analysis, from the geometry modelling until the determination of the strain from the interface.
There are totally two FE models that need to be modelled and analyzed in this study. The first model is Reinforced Concrete beam
Before starting to define this or any model, we need to decide which system of units we will use. ABAQUS has no built-in system of units. All input data must be specified in consistent units. Some common systems of consistent units are shown in Table 4.1. In entire this thesis followed units are in SI units in millimeters.
Table 4.1 Consistent units

Quantity	SI	SI (mm)	US Unit (ft)	US Unit (inch)
Length	m	mm	ft	in
Force	N	N	lbf	lbf
Mass	kg	tonne (10^3 kg)	slug	lbf s^2/in
Time	s	s	s	s
Stress	Pa (N/m^2)	MPa (N/mm^2)	lbf/ft^2	psi (lbf/in^2)
Energy	J	mJ (10^{-3} J)	ft lbf	in lbf
Density	kg/m^3	tonne/mm^3	slug/ft^3	lbf s^2/in^4

4.7 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 4.1 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.

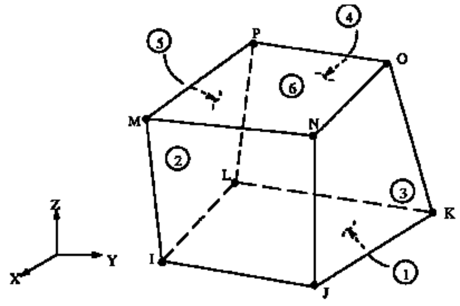


Figure 4.1: C3D8R-3D solid element (ABAQUS)
The solid element has eight nodes with three degrees of freedom at each node – translations in the nodal x, y, and z directions. The element is capable of plastic deformation, cracking in three orthogonal directions, and crushing. Necessary partitions of the concrete beam (of size 1200 x200 x100) are made to facilitate load application and meshing as shown in the Figure 4.2

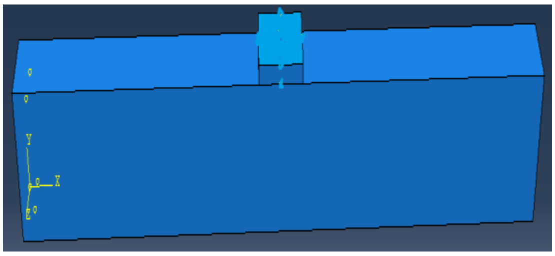


Figure 4.2: Developed solid model
The steel reinforcement of size 1150 mm is modeled as two –node beam elements connected to the nodes of adjacent solid elements. Each node has three degrees of freedom, – translations in the nodal x, y, and z directions. The element is also capable of plastic deformation. The geometry and node locations for this element type are shown in Figure 4.3

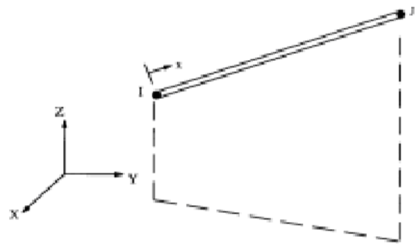


Figure 4.3: B31-3D Spar (ABAQUS)
On the basis of the sections available in the ABAQUS commercial package library, any profile can be assigned by regarding their shape and dimensions. The orientation of the beam cross section is defined which is available default in ABAQUS. For simplicity, stirrups, which are used to ensure beams do not fail in shear during experiment, were not considered in the simulations. Figure 4.4. indicates the section chosen to develop flexural reinforcement.

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

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

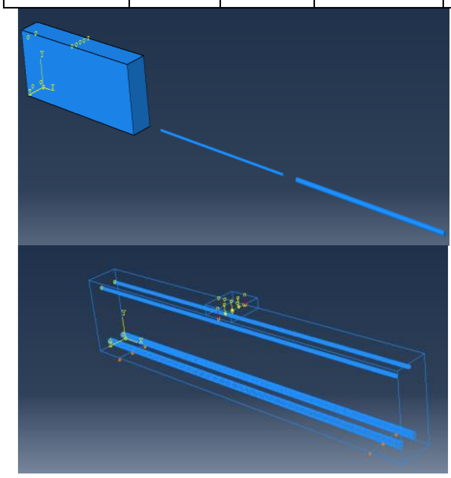


Figure 4.9 Assembling of part instances

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 4.9 shows assemblage of all parts instances

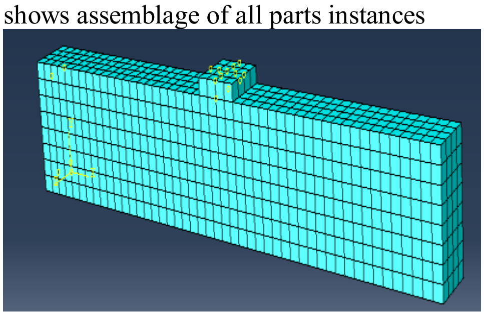


Figure 4.15 Meshing of concrete beam

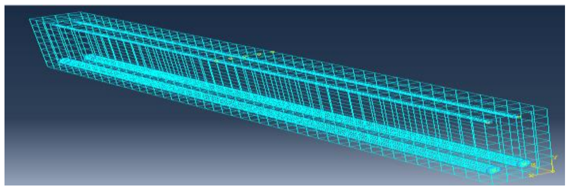


Figure 4.16 Meshing of steel bars

Table 4.2: Details of mesh

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 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.

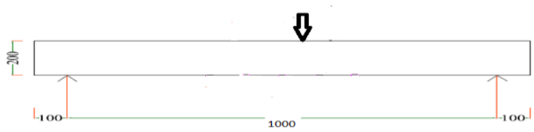


Fig.5.1 Testing of Plane/Reinforced concrete beam by three point loading

Fig: 5.2 Flexure Failure of under reinforced concrete beam

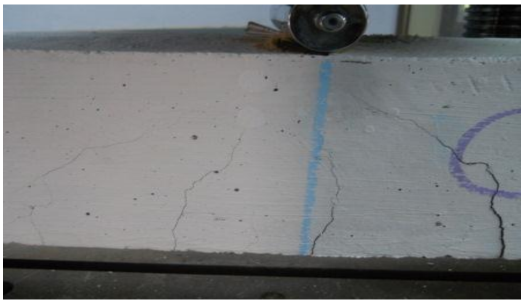


Fig: 5.3 Flexure Failure of over reinforced concrete beam


Fig: 5.4 Flexure Failure of over/balanced/under reinforced concrete beam

Name of Section	Deflection		stress in concrete		stress in steel		1st Crack at kn	Ultimate load	
	FEA	Exper	Tension	compr	Tension	crack		FEA	Exper
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

Fig: 5.5 Results of plane/over/balanced/under reinforced concrete beam

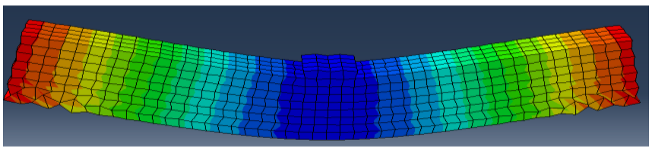


Fig: 5.6 Deformation of Plane concrete beam

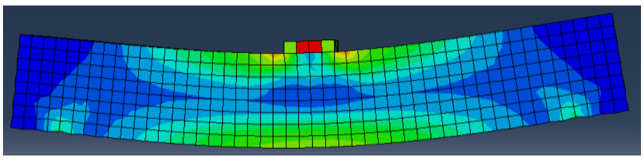


Fig: 5.7 (a) Stress in plane concrete beam

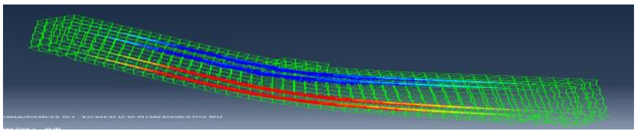


Fig: 5.7 (b) Stress in steel bar in under reinforced concrete beam

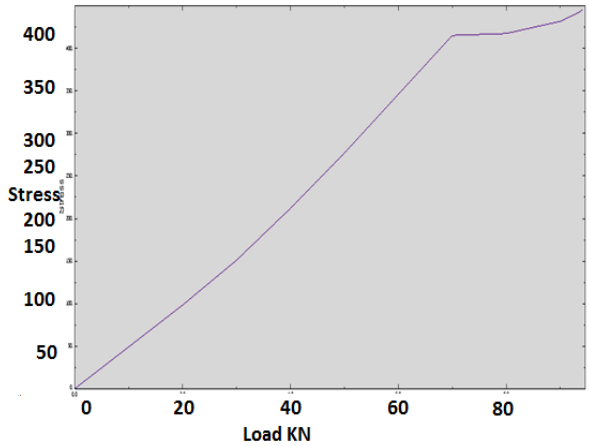


Fig: 5.8 Stress in steel node

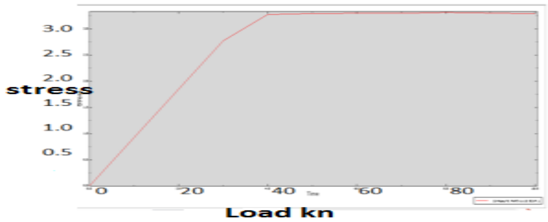


Fig: 5.9 Stress concrete node

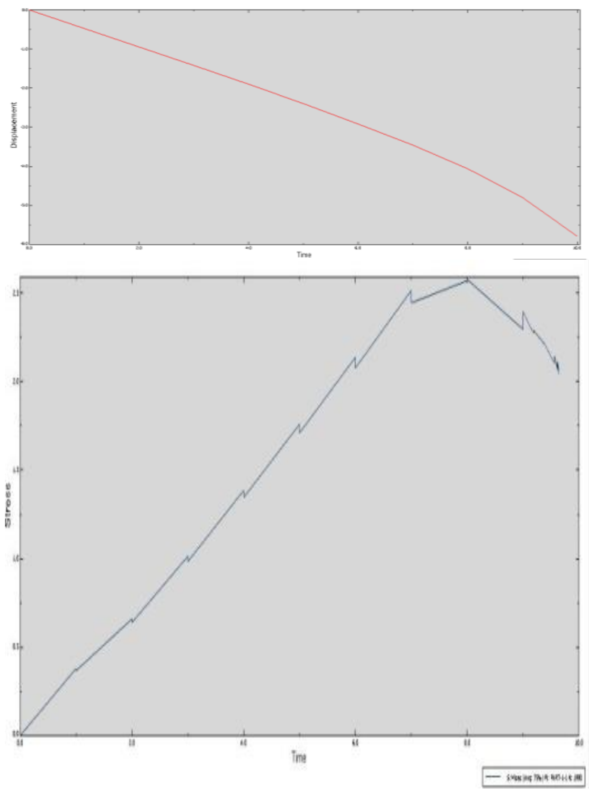


Fig: 5.10 Stress in steel, concrete of under reinforced beam section

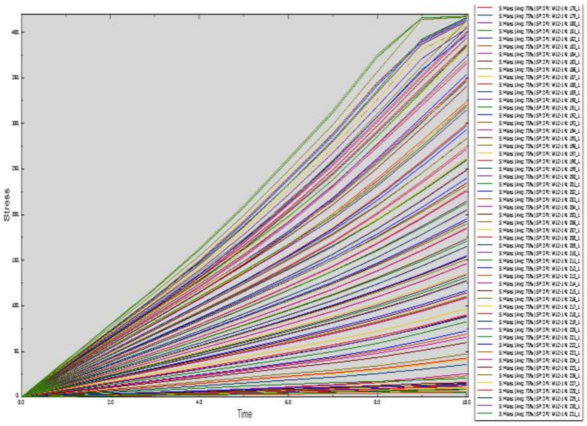


Fig: 5.11 Stress in plane concrete beam section

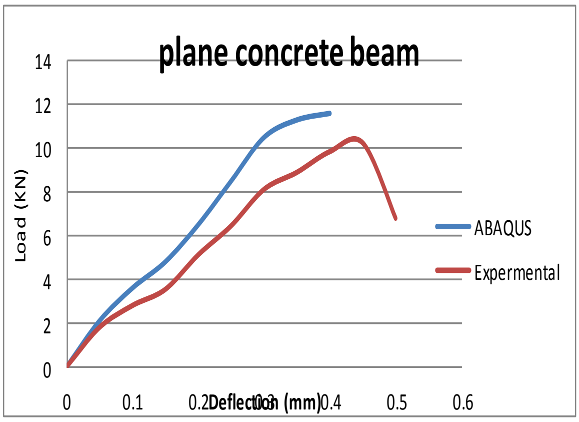
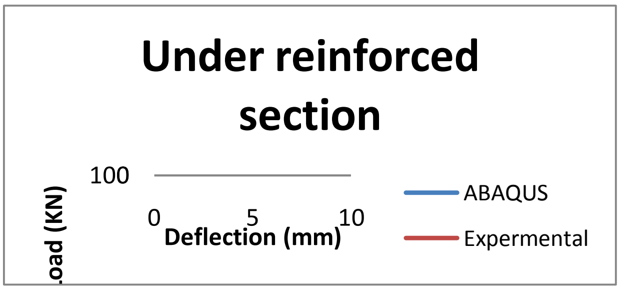


Fig: 5.12 Deflection graph time(load) vs Deflection



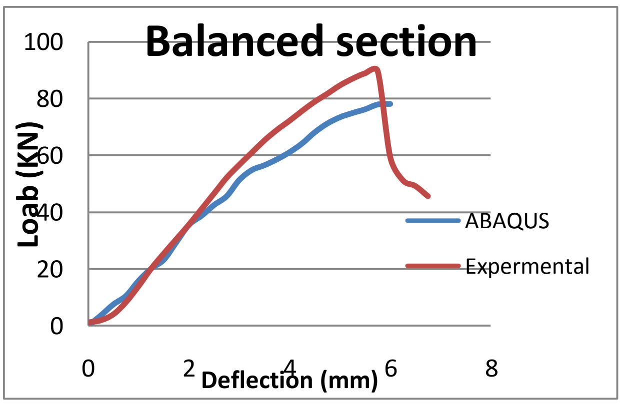
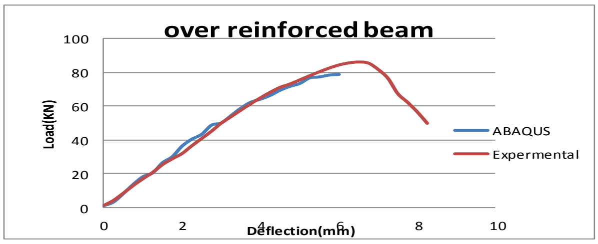
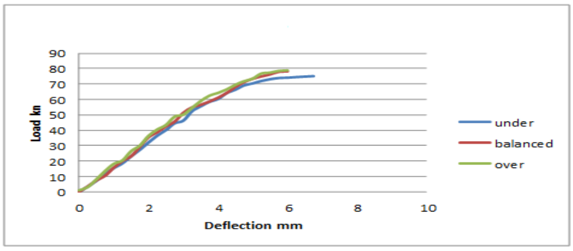
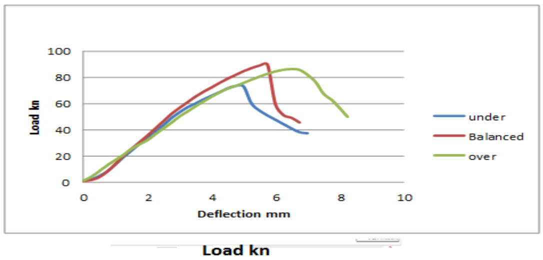


Fig: 5.13 Stress in steel bar at each node





Comparison of over/balanced/under reinforced beams from ABAQUS



Comparison of over/balanced/under reinforced beams from Experimental

Conclusion

The following conclusions can be stated based on the evaluation of the analyses of the calibration model .Deflections and stresses at the centreline along with initial and progressive cracking of the finite element model compare well to experimental data obtained from a reinforced concrete beam. The ultimate load carrying capacity of Plane concrete beam is 0.14 times under reinforced beam The failure mechanism of a reinforced concrete beam is modelled quite well using FEA and the failure load predicted is very close to the failure load measured during experimental testing. In under reinforced beam maximum elements reach ultimate stress compare to over reinforced concrete beam 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. 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.

References

1. Saifullah M.A. Hossain Experimental and Analytical Investigation of Flexural Behavior of Reinforced Concrete Beam
2. Anthony J. Wolanski, B.S. (2004) “Flexural Behavior of Reinforced and Prestressed Concrete Beams Using Finite Element Analysis”, Master’s Thesis, Marquette University Milwaukee, Wisconsin.

3. Vasudevan. G 1 , Kothandaraman.S 2 Parametric study on Nonlinear Finite Element Analysis on flexural behaviour of RC beams using ANSYS
4. R. Srinivasan and K. Sathiya flexural Behavior of reinforced concrete beam using finite element analysis Concrete structural components exist in buildings in different for
5. Hsuan-Teh Hu *, Fu-Ming Lin, Yih-Yuan Nonlinear finite element analysis of reinforced concrete beams strengthened by fiber-reinforced plastics
6. Hamid Sinaei*, Mahdi Shariati, Amir Hosein Abna, Mohammad Aghaei and Ali Shariati Evaluation of reinforced concrete beam behaviour using finite element analysis by ABAQUS
7. Mohammed Sh. Mahmood Finite Element Modeling of Reinforced Concrete Beams Strengthened with FRP Laminates
8. 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.
9. R. Balamuralikrishnan1,* andC. Antony Jeyasehar Flexural Behavior of RC Beams Strengthened with Carbon Fiber Reinforced Polymer (CFRP) Fabrics
10. Kasidit Chansawat.et.al., FE Models of GFRP and CFRP Strengthening of Reinforced Concrete Beams. *Submitted to Advances in Civil Engineering.*
11. Nilson, A. H., Darwin, D., and Dolan C. W.,Edition, (2006), "Design of Concrete Structure", McGraw-Hill Education(Asia),Singapore.
12. 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.
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. Nakasone, Y.; Yoshimoto, S.; Stolarski, T. A., 2006, "ENGINEERING ANALYSIS WITH ANSYS SOFTWARE", ELSEVIER, 1st Published.
16. Hossain. Nadim, M, 1998, "Structural Concrete Theory & Design", Addison-Wesley Publishing Company
17. Altenbach.H, Altenbach.J & Kissing.W, (2004). Mechanics of Composite Structural Elements. Ref No: ISBN 3-540-40865-7
18. Taljsten, B. (1997). Strengthening of Beams by Plate Bonding. *Journal of Materials in Civil Engineering,* 9, 4, 206-12.
19. 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,*
20. 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.
21. 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.
22. Vecchio, F.J., and Bucci, F. (1999). Analysis of Repaired Reinforced Concrete Structures. *Journal of StructuralEngineering,*125,6,644-52