Analysis of RCC Beams using ABAQUS

T. Tejaswini
PG Student, Dept. of Civil Engineering
CBIT, Hyderabad, Telangana, India

Dr.M.V.Rama Raju
Assoc. Professor, Dept. Of Civil Engineering
CBIT, Hyderabad, Telangana, India

Abstract - Reinforced concrete (RC) has become one of the most important building materials and is widely used in many types of engineering structures. For the efficient use of RCC it is necessary to know the properties and the behavior of RCC elements under various constrains. 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 valuable in the preliminary design stages. Experimental research also supplies the basic information for finite element models. The development of reliable analytical models can 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 aim of the present study is to compare experimental results with the ABAQUS results. Initially Laboratory tests are carried out on a beam of 1200 x 200 x100 mm of M30 grade concrete for plain, under, balanced, over reinforced sections. Finite Element Analysis (FEA) have also been performed using ABAQUS for the model geometry considered in the experimental study. The numerical results from the FEA are compared with the experimental results which showed good agreement between the results.

Keywords – Beam, Beam sections, ABAQUS, Analytical modelling,

I. INTRODUCTION

A concrete beam is a structural element that carries load primarily in bending. 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 analyze 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 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.

II. DESIGN CRITERIA

Concrete structure must satisfy the following conditions: (1) The structure must be strong and safe (2) The structure must be stiff and appear unblemished (3) The structure must be economical.
Advanced analytical tools can be an indispensable aid in the assessment of the safety and the serviceability of a proposed design. 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.
For the analysis of structural member, ABAQUS has been chosen for the purpose of modeling and analyzing the concrete beam with steel in this study due to its flexibility in creating geometry and material modeling.

III. DESCRIPTION OF SPECIMENS

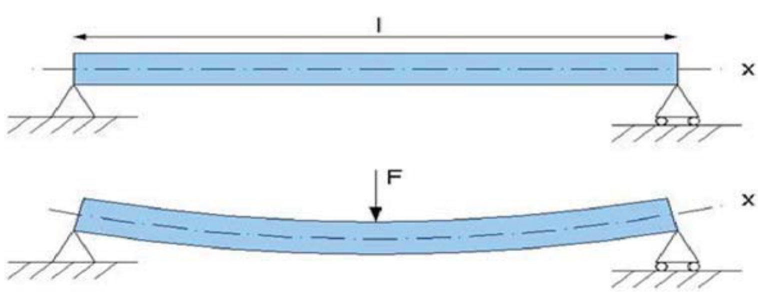
Two plain concrete beams, Six Reinforced concrete beams of M30 grade OPC concrete (1:1.34:2.88) used in this investigation program were it 100 mm wide x 200 mm deep x 1200 mm long. Two beams (Under reinforced) are casted with 2- 12Ø bars at bottom and 2-8Ø hanger bars, 8Ø @135 stirrups are provide, And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

Table 1. Details of beam 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	……….

IV. 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.



Procedure:

To analyze the relationship between load and deflection of a beam, Universal testing machine is used with beam placed on two supports affected by a concentrated load at the center.

1. Set the bearers so that a span of 1200 mm is maintained. The interval between each groove on the beam of the apparatus is 100 mm.
2. Place a test specimen with dimensions of 100 x 200 mm. on the bearers and mount the load device in the center of test specimen.
3. Set the testing device so that the top of the gauge is centered on the upper plane of the load device. Lower the gauge so that its small hand is at about 10 and set the gauge to zero by twisting its outer ring. Load with weights as shown in the table below and read off the deflection. One revolution of the large hand of the gauge corresponds to 1 mm. of deflection.

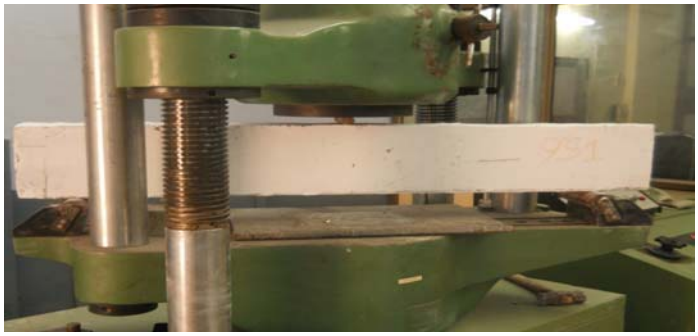


Figure 1. Three point loading test using UTM machine

Figure 2. Flexure Failure of over/balanced/under reinforced concrete beam

Experimental Results for various Beam Sections:

Table2. Load and corresponding deflections for a balanced section

Load (kN)	Deflection (mm)
20	1.25
40	2.25
60	3.2
80	4.65
88.8 (ultimate)	5.75

Table3. Load and corresponding deflection for an under reinforced section

Load (kN)	Deflection (mm)
20	1.15
40	2.35
60	3.56
70	4.65
73.3 (ultimate)	5.2

Table4. Load and corresponding deflection for an over reinforced section

Load (kN)	Deflection (mm)
20	1.25
40	2.85
60	3.95
80	5.45
86.2 (ultimate)	5.8

V. ABAQUS MODELLING

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

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.

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.

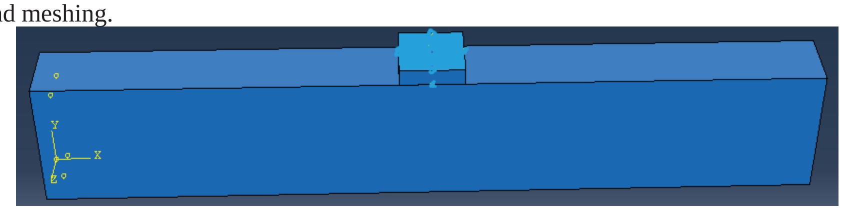
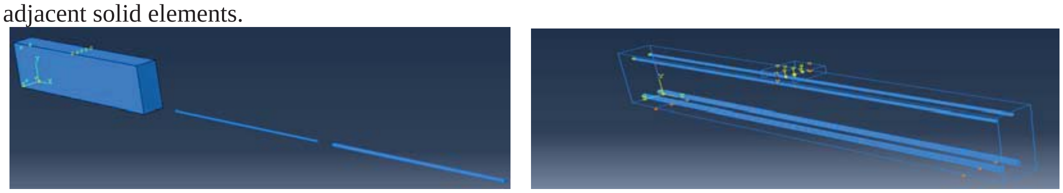


Figure 3. Developed solid model

The steel reinforcement of size 1150 mm is modeled as two –node beam elements connected to the nodes of adjacent solid elements.



(a) (b)

Figure 4. Assembling of part instances

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. The beam is simply supported, all the nodes at a distance of 100 mm from both the edge of the beam is retrained to move along Y direction at one side and on the other side it is restrained to move in X and Y direction as shown in Figure.

Meshing is the process of generating nodes and elements. A mesh is generated by defining nodes and connecting them to define the elements

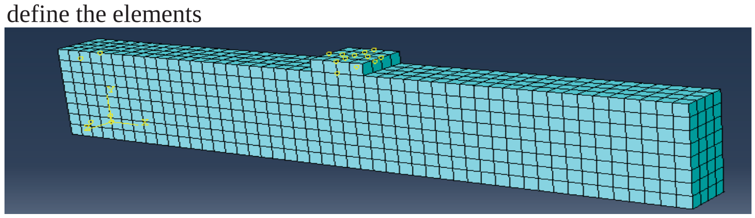


Figure 5. Meshing of concrete beam

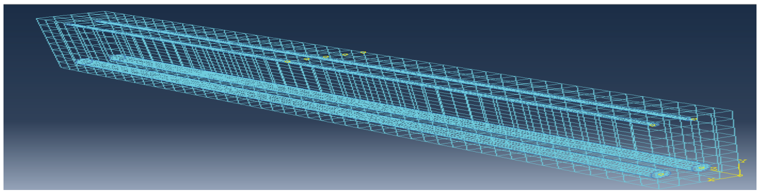


Figure 6. Meshing of steel bars

Creating analysis job

In order to solve any type of finite element problem, the relevant job analysis should be established. After this stage the extracted answers is visualized analytically and graphically.

Viewing results

Finally when the solution is complete, the results can be viewed. These results may be of color contour plots, line plots, or simply lists of degrees of freedom for each node. It is analysis phase where the results of analysis are reviewed through graphics and graph.

VI. RESULTS AND DISCUSSION

Name of	Deflection		stress in concrete		stress in steel		1 st	Ultimate load	
Section	FEA	Expr	Tension	Comp	Tension	comp	Crack at kN	FEA	Expr
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

Table5. Comparison of FEA and Experimental results for different sections.

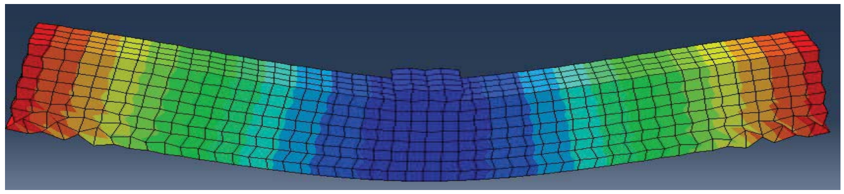


Figure 7. Deformation of Plane concrete beam

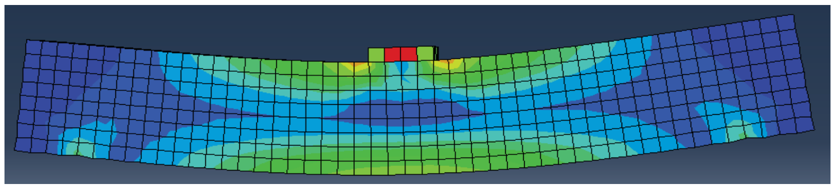


Figure 8. Stress in plane concrete beam

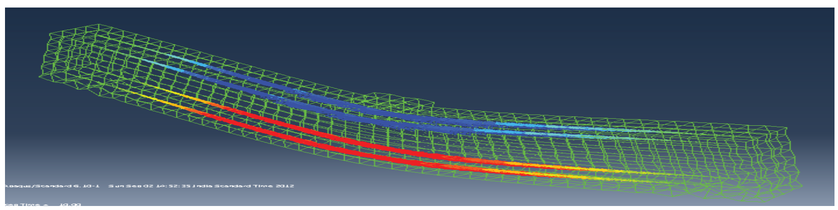


Figure 9. Stress in steel bar in under reinforced concrete beam

Figure 7. Shows the deformation pattern of a plane concrete beam where maximum deformation is observed at the center of the beam under the loading. Figure 8, 9 shows the stresses at various points of the beam section in a plane concrete beam and the steel bars in the reinforced concrete beams.

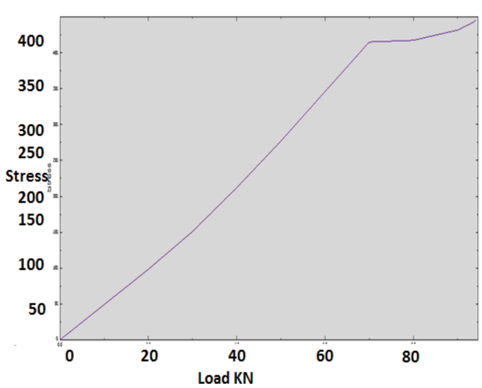


Figure 10. Stress in steel node

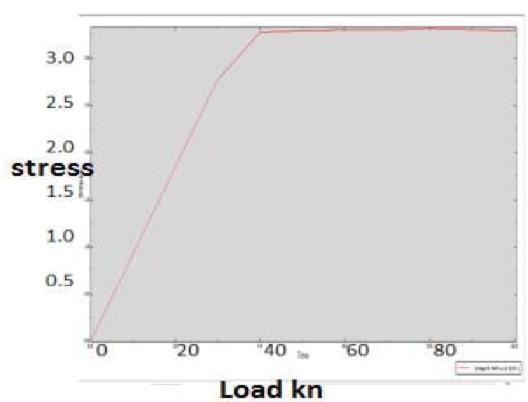
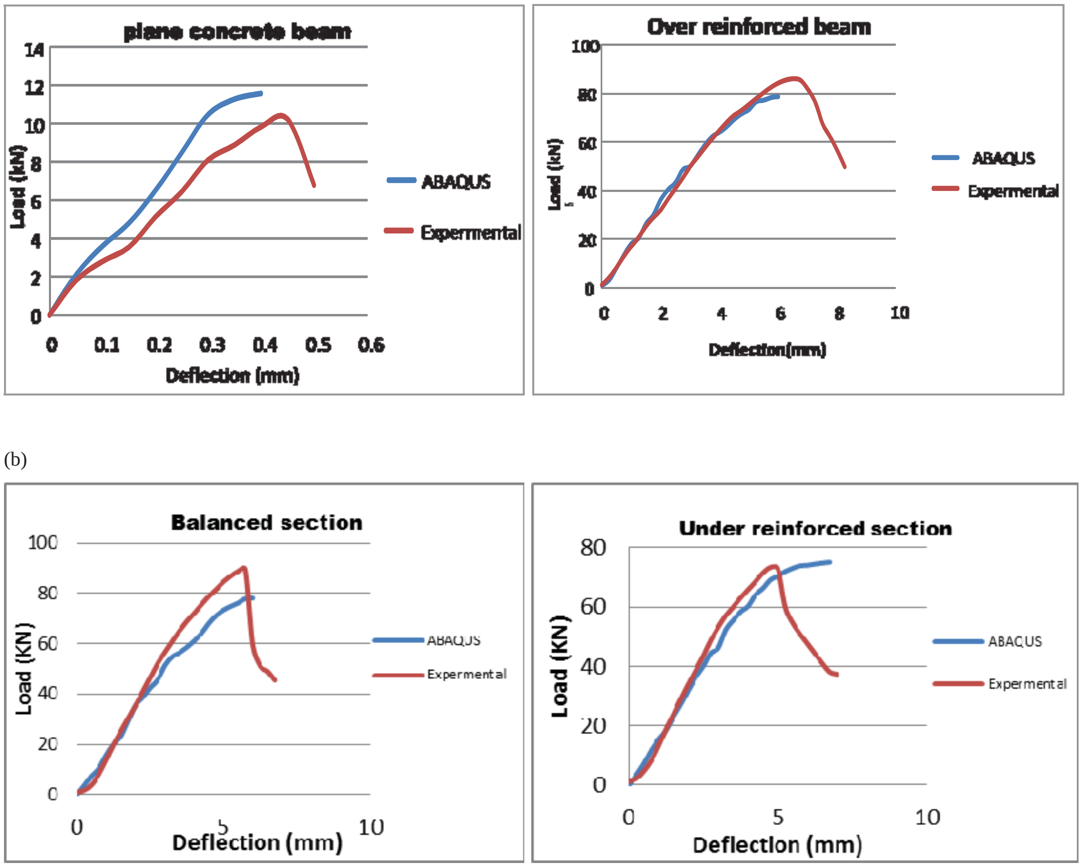


Figure 11. Stress concrete node

The figures 10, 11 shows the stress variation in steel and concrete nodes for increasing load.



(a)

(b)

(c) (d)

Figure 12. Load deflection behavior of various sections

The ABAQUS produced a load deflection behavior which deviates slightly from the experimental curve in case of plane concrete beam. It can be observed that the correlation between experimental results and analytical results are quite good.

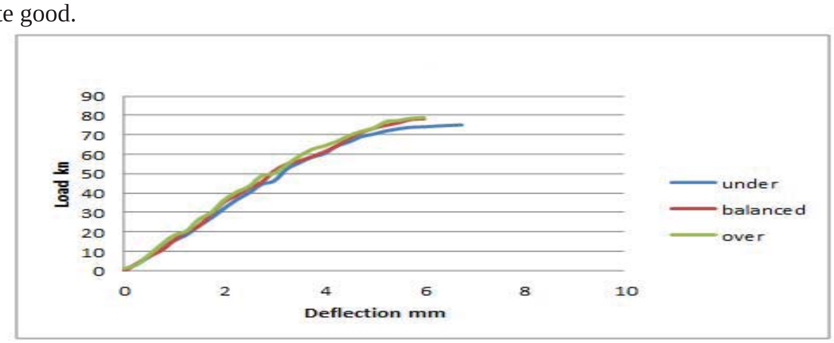


Figure 13. Comparison of over/balanced/under reinforced beams from ABAQUS.

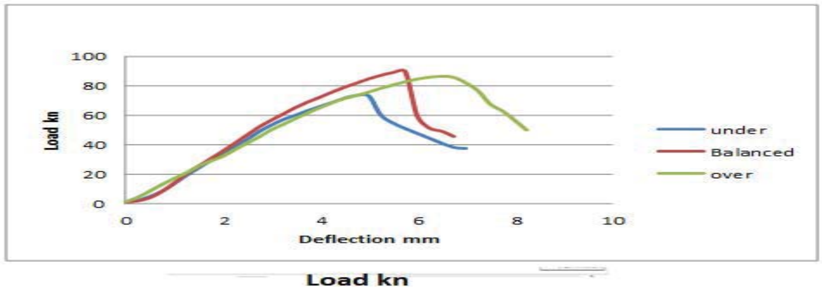


Figure 14. Comparison of over/balanced/under reinforced beams from Experimental

From experimental results it is seen that the under reinforced concrete beam shows greater warning zone before failure.

VII. 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. 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.
5. In under reinforced beam maximum elements reach ultimate stress compare to over reinforced concrete beam.

REFERENCES

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

[2] Nitereka, C., and Neale, K.W. (1999). Analysis of Reinforced Concrete Beams Strengthened in Flexure with Composite Laminates.

[3] 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. *Canadian Journal of Civil Engineering*, 646-54.

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

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

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

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

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

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

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

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

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

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

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

[15] 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

[16] Altenbach.H, Altenbach.J & Kissing.W, (2004). Mechanics of Composite StructuralElements. Ref No: ISBN 3-540-4086

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