IJCRT.ORG ISSN : 2320-2882
INTERNATIONAL JOURNAL OF CREATIVE RESEARCH THOUGHTS (IJCRT)
An International Open Access, Peer-reviewed, Refereed Journal

FINITE ELEMENT MODELING OF RCC BEAM BY USING ABAQUS

T.S.Vishnu Kumar[1], N.Rushwanth Chowdary[1]

1. Academic Consultant, Dept. of Civil Engineering, SV University College of Engineering, Tirupati.

ABSTRACT

RCC (Reinforced Cement Concrete) is one of the widely used materials in the construction sector in India. It is necessary to understand the behavior of RCC member to check the stability aspect of the structure. Therefore, the development of a finite element model (FEM) may need intensive material testing to incorporate into the material model in any of the finite element (FE) packages available (Sinaei et al., 2011). There are quite a large number of numerical material models available in the literature with the potential to develop complete stress-strain curves of concrete for compression and tension separately based on the experimental results. In this research, the ABAQUS program Hibbitt et al. (1988) is used to model the behavior of RCC beams.

Keywords: - ABAQUS, RCC Beam, FEM

I. INTRODUCTION

Reinforced concrete is a complicated material to be modelled within finite element packages. A proper material model in the finite element model should inevitably be capable of representing both the elastic and plastic behaviour of concrete in compression and tension. The complete compressive behaviour should include both elastic and inelastic behaviour of concrete including strain softening regimes. The simulation of proper behaviour under tension should include tension softening, tension stiffening and local bond effects in reinforced concrete elements. The finite element model uses the concrete damaged plasticity approach; this model can help to confirm the theoretical calculations as well as to provide a valuable supplement to the laboratory investigations of behaviour. For validation, a reinforced concrete beam was modelled which had been experimentally tested and reported by Kachlakevetal. (2001).

The primary concern about the structural failure under anticipated extreme loadings had been addressed with the many provisions attributed to issues like: (i) attaining member level/structure level resistance against the code prescribed loads and load combinations (ii) defining all anticipated loads including extreme environmental loads. (iii) Attaining structural integrity and ductility. (iv) providing continuity for connections and (v) providing some general statements about resiliency, redundancy and robustness, etc. Therefore, code-compliant buildings highlight the significance of more investigative study related to impact loading issues. It is recognized that experimental observations are much more expensive comparative to its counterpart investigations implementing computer analyses. In that sense, finite element (FE) analysis can be an option. Thus, a 3D FE analysis model using ABAQUS was triggered to explicitly explore the dynamic behavior of a reinforced concrete (RC) structural element. ABAQUS is a very complex FE analysis program introduced with huge material characteristics and parameters to reproduce high accuracy in calculations. But there are no clues to assign exact values for those parameters. Thus, model choosing based on influential material characteristics and behavioral parameters is an important part of FE analysis before investigating the actual behavior of a structural element. In this study, thirty analyses have been executed changing different parameters, such as damping, tension and compression stiffness recovery, damage parameter-strain/ displacement relations and friction coefficient to choose the best performing FE model. The results of few analyses are to be discussed in this paper. Upon extensive examination of the calculated structural responses of the FE models comparing with the published experimental results. This study reveals that FE analysis using proposed model can be applied to explore dynamic behavior of structural elements subjected to impact vibrations

II. LITERATURE REVIEW

Duthinh and starnes, (2001) conducted experiment on strengthening of reinforced concrete beams using carbon fiber reinforced polymer. The seven test beams were cast and strengthening externally with carbon fiber reinforced polymer (FRP) laminated after the concrete had cracked were tested under four point bending. The results obtained from this experiment were that CFRP is very effective for flexural strengthening. As the amount of steel reinforcement increases, the additional strength provided by carbon FRP external reinforcement decreases. The same FRP reinforcement more than doubled the strength of a lightly reinforced beam. Compared to a beam reinforced heavily with steel only, the beams reinforced with both steel and carbon have adequate deformation capacity, in spite of their brittle mode of failure.

In the present model, a solid foundation was used, which is more realistic although this modelling requires much more memory for the resolution. The solid foundation is more realistic than the liquid foundation, because the deflection in any nodal point depends not only on the force in this node but also of the forces in all the other nodes. Moreover, all the analytical solutions of the 2D models are based on the proposal that the slab and the foundation are in perfect contact (Coquand, 1989). With the advantages of calculations of the numerical methods by computers, the developed analyses are based on a partial contact between the layers (Huang, 2004). This work includes, mainly, a 3D modelling by the use of the computer code of the finite elements "Abaqus 6.7" in order to understand, with more precision, the distribution and the evolution of the stresses and displacements in the entire RCC slab. These results were shown especially for many loading position, in top and bottom fibres and in the interface with the sub-base. In the numerical 3D modeling by finite elements using Abaqus 6.7, the authors choose various options of manual and automatic incrementing with an automatic tolerance of convergence. The control parameters of the management of the numerical analysis of the problem are adjusted automatically with a low manual adjustment. The convergence criteria are also adjusted during the analysis to ensure a precise solution. In this numerical approach, two RCC slab separated by a joint, were modelled. Both rest on a gravel sub-base suitably compacted. The whole also rests on a ground support. The geometries and the mechanical properties of material were introduced. These introduced parameters were: the elastic modulus E, the Poisson's ratio n and the admissible stresses of tensile and compression. All dimensions are finite in 3D; the diagrams of the model are presented.

III. DESCRIPTION OF THE SPECIMEN

The theoretical Finite Element Analysis, the computation of membrane stresses and deflections corresponding to the applied loads on RCC Beam will be carried out using the standard program (ABAQUS Version 6.10).

DIMENSION OF THE SECTION

Length - 2.0m

Width - 0.35m

Depth - 0.45m

Square mesh 2mm diameter

Main Steel bars 8mm diameter& Stirrups 6mm diameter.

Clear cover – 25mm

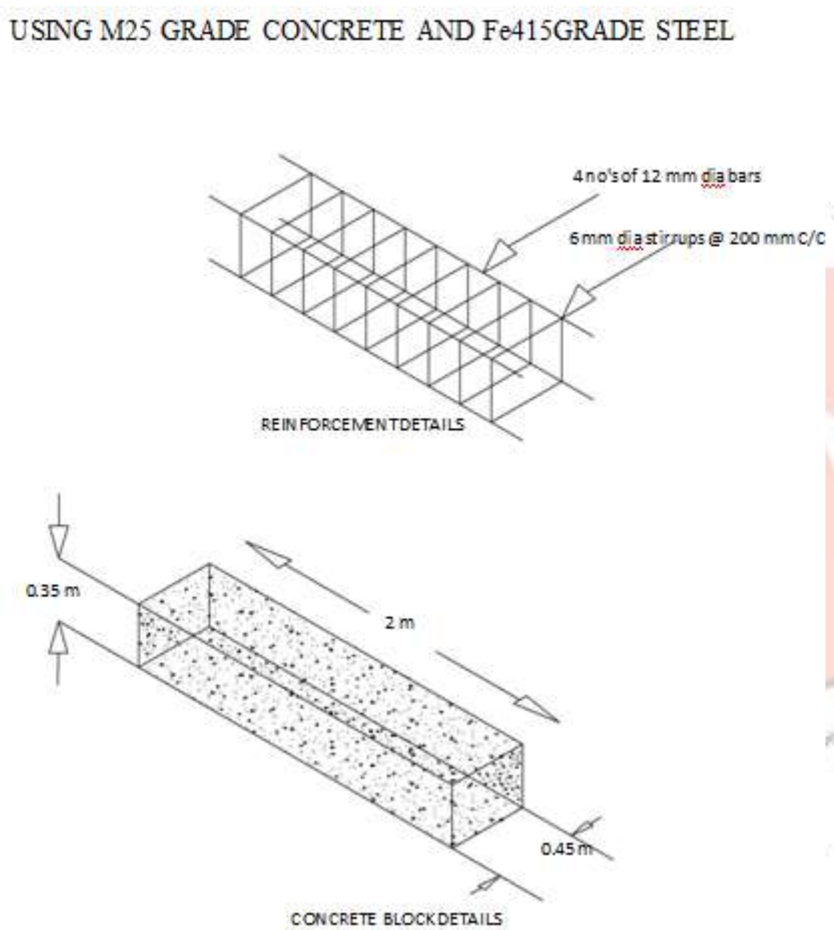
EXECUTION OF PROGRAM

The analysis and the computation of RCC Beam are done using standard program, by executing in the following steps.

- Two models are created for RCC Beam in ABAQUS.
- First created the concrete model.
- Steel model created by used co-ordinates and by used wire option to connected each node by node.
- A material property is assigned as elastic properties.
- Section is assigned as homogenous.
- Mesh and steel model are assigned as solid model.
- Assembled the two models by used constrains into TIE option

- Job was input
- Linear analysis was done.
- Deflection and stress contour is noted.

AUTOCAD MODEL FOR RCC BEAM



- Figure 1. AUTOCADD model of RCC Beam

IV. SIMULATIONS

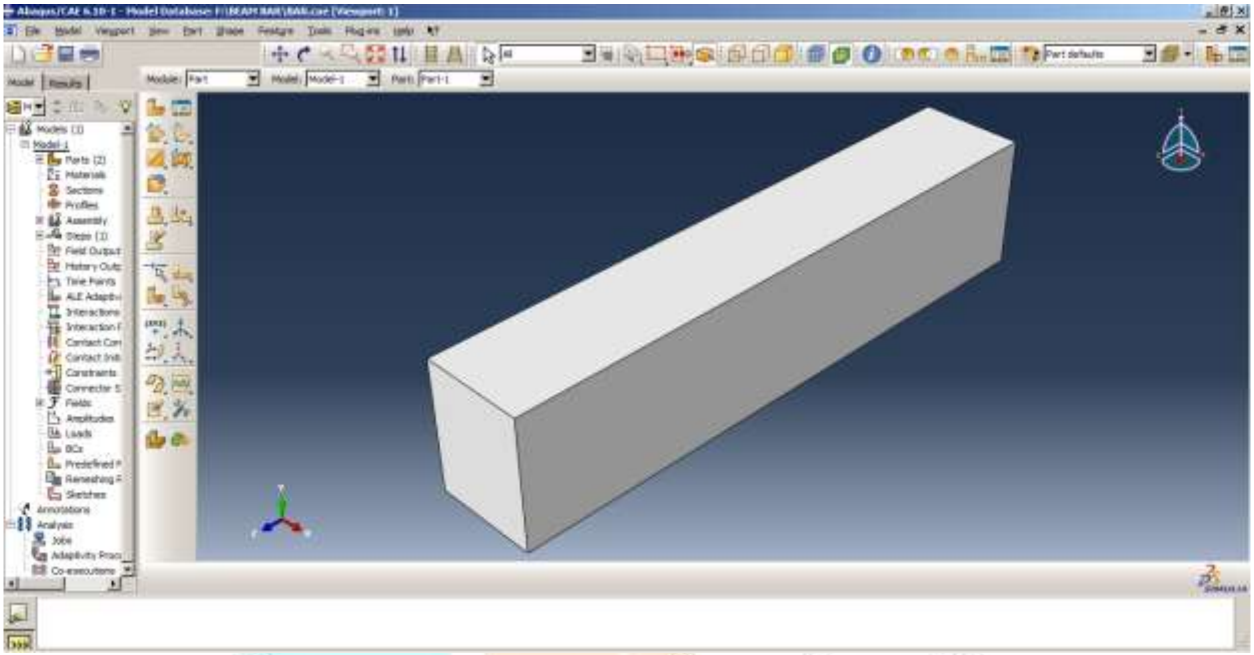


Figure 2. Concrete model in ABAQUS

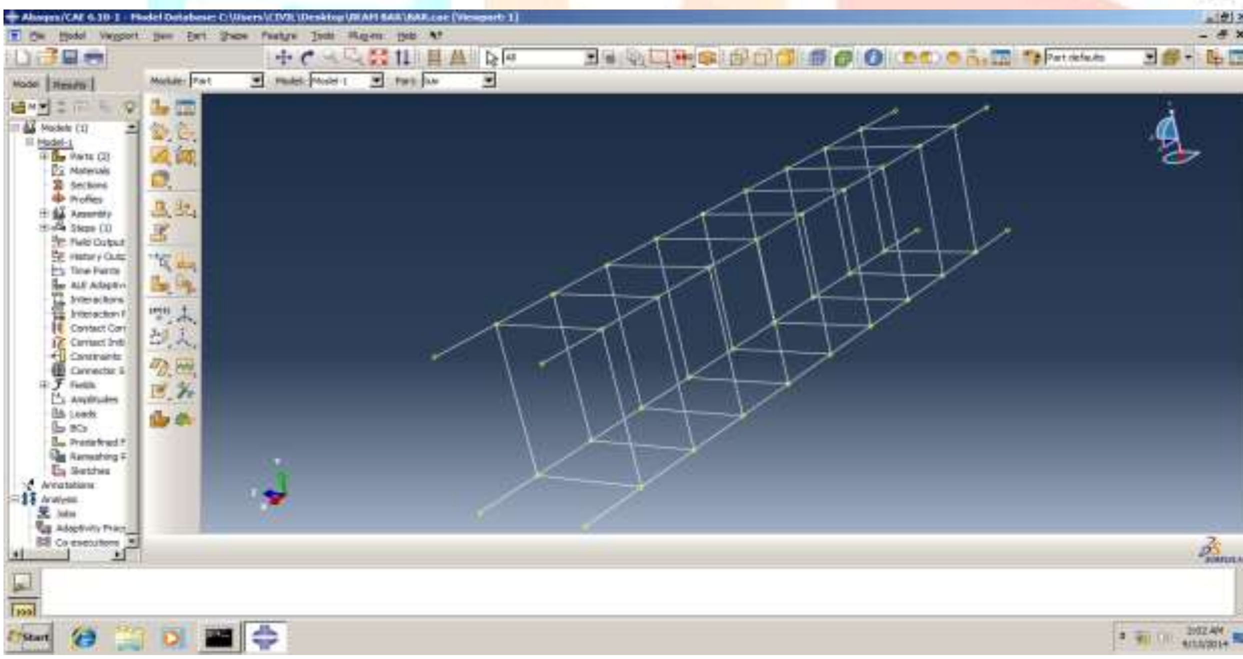


Figure 3. Steel model in ABAQUS.

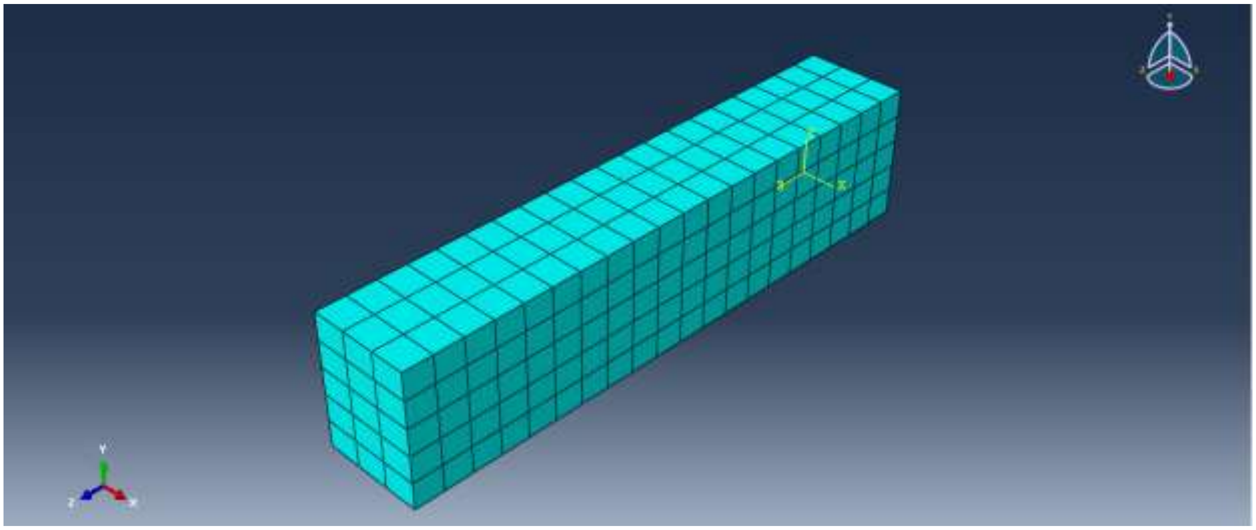


Figure 4. Assembled model for RCC Beam

MATERIAL PROPERTIES

The material properties are taken from **IS 456:2000**

Elastic modulus of concrete = $5000\sqrt{f_{ck}}$

$= 5000\sqrt{25}$

$= 25000$ N/mm^2

Table 1. Young's modulus and Poisson's ratio for mesh, steel and mortar.

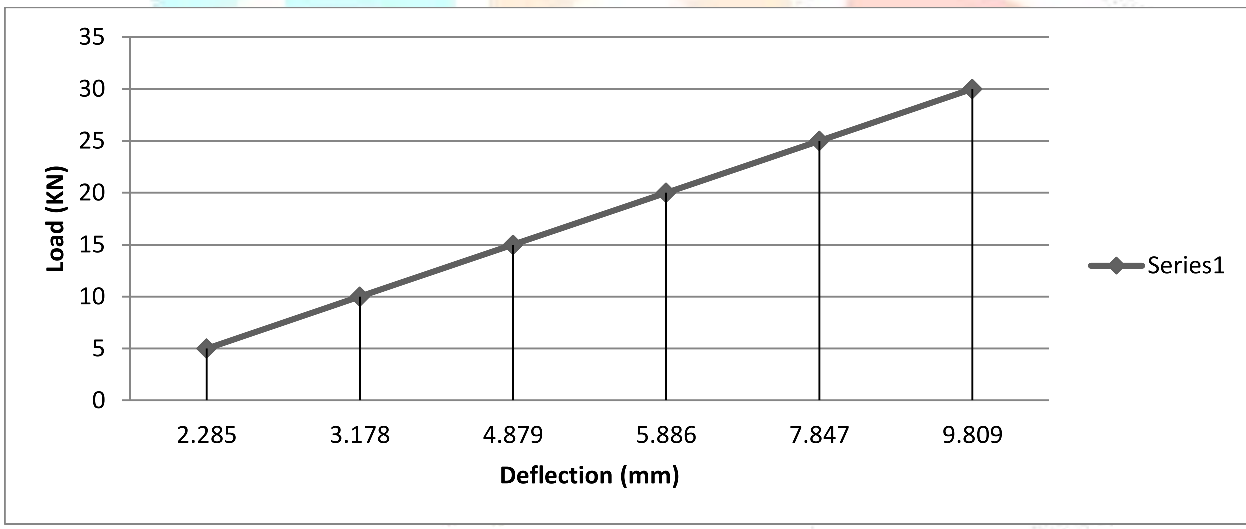
S.no	**Material's**	**Young's modulus E**	**Poisson's ratio υ**
1	Cement concrete	25 Gpa	0.2
2	Square mesh	200 Gpa	0.3
3	Steel	200 Gpa	0.3

The analysis for the RCC Beam is done in ABAQUS by FEM meshing tool. FEM meshing is done by quadrilateral mesh. Boundary condition is assigned as fixed condition. Concentrated load is applied on the specimen. Job was input. Linear analysis was done.

V. RESULTS

POINT LOAD AND CORRESPONDING RESULTS

S.No	Load in kN	Deflection in mm
1	5	2.285
2	10	3.178
3	15	4.879
4	20	5.886
5	25	7.847
6	30	9.809



Graph 1. Load vs. Deflection

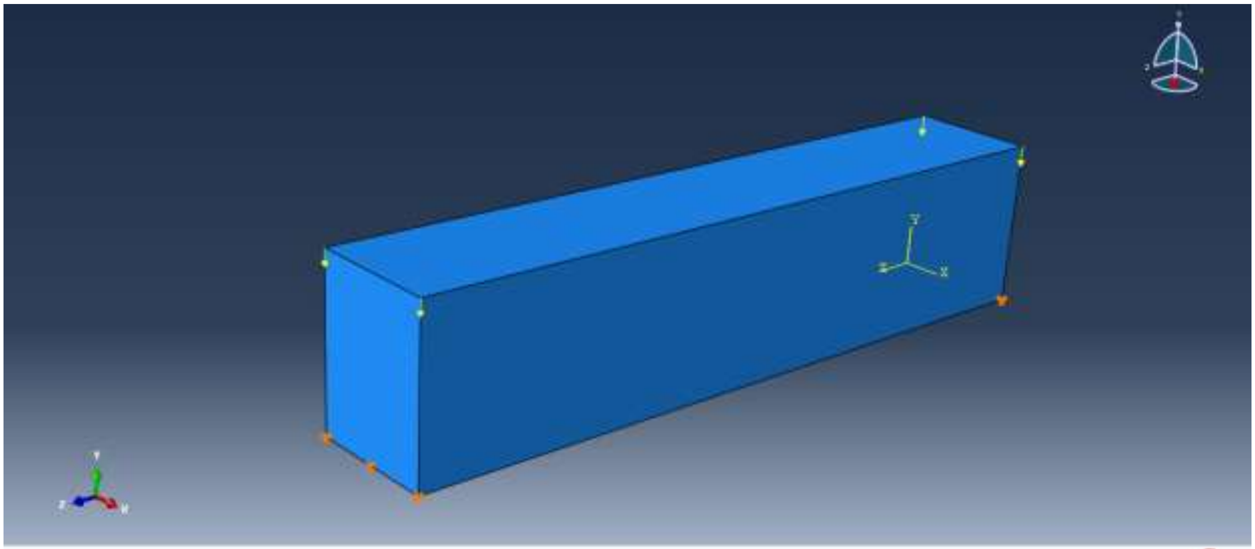


Figure 5. simply supported beam with point load

The discretization is done for the clear representation of the contour. Stress contour and deflection are shown comparing the top to bottom sides

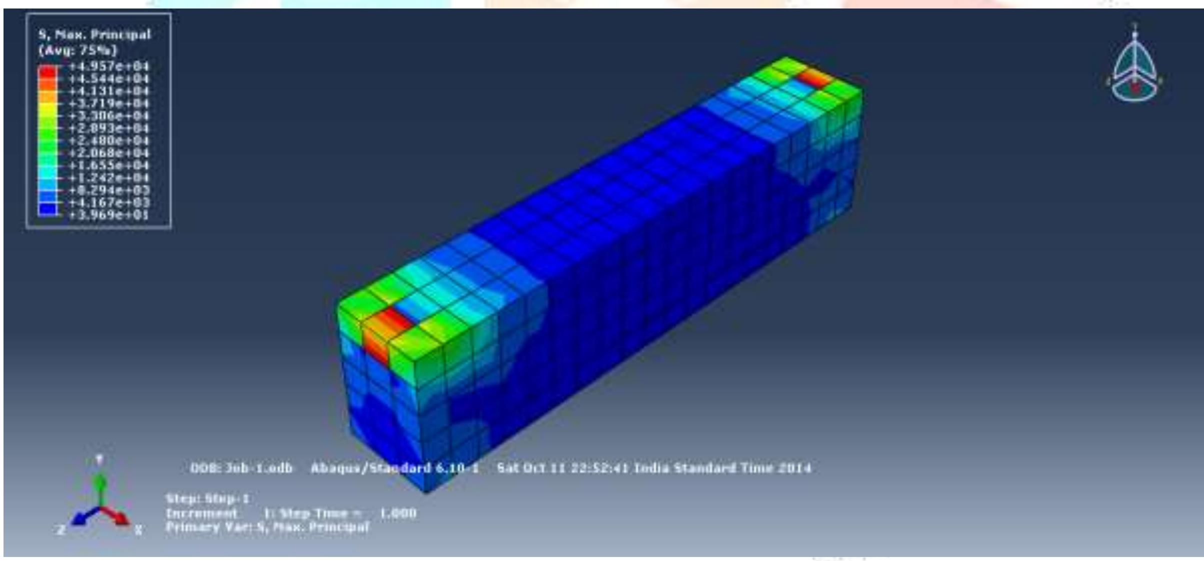


Figure 6. Maximum principal stress

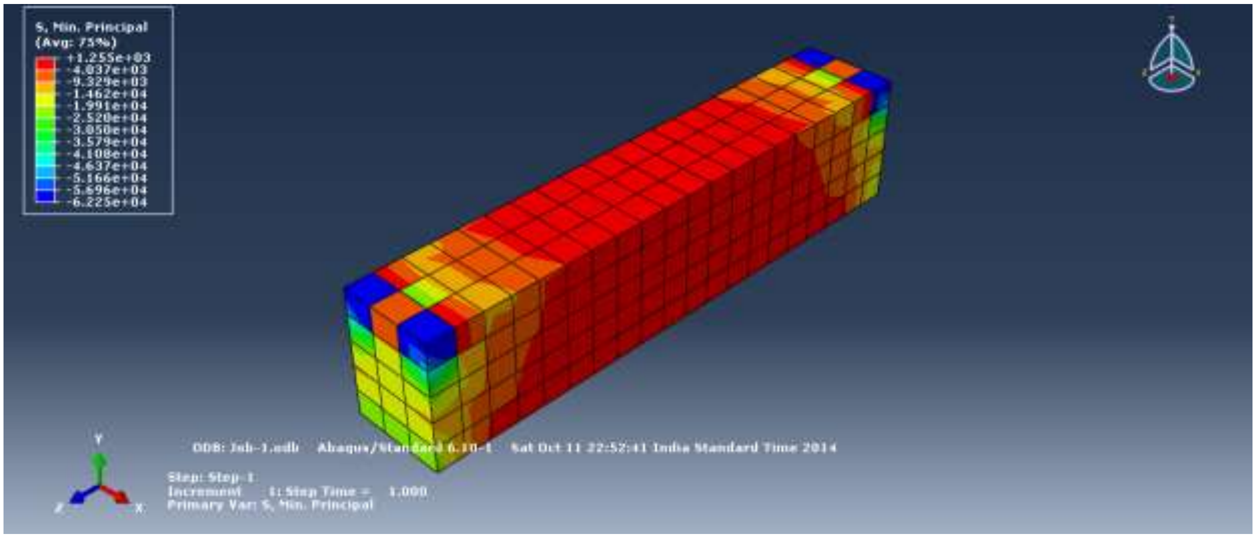


Figure 7. Minimum principal stress

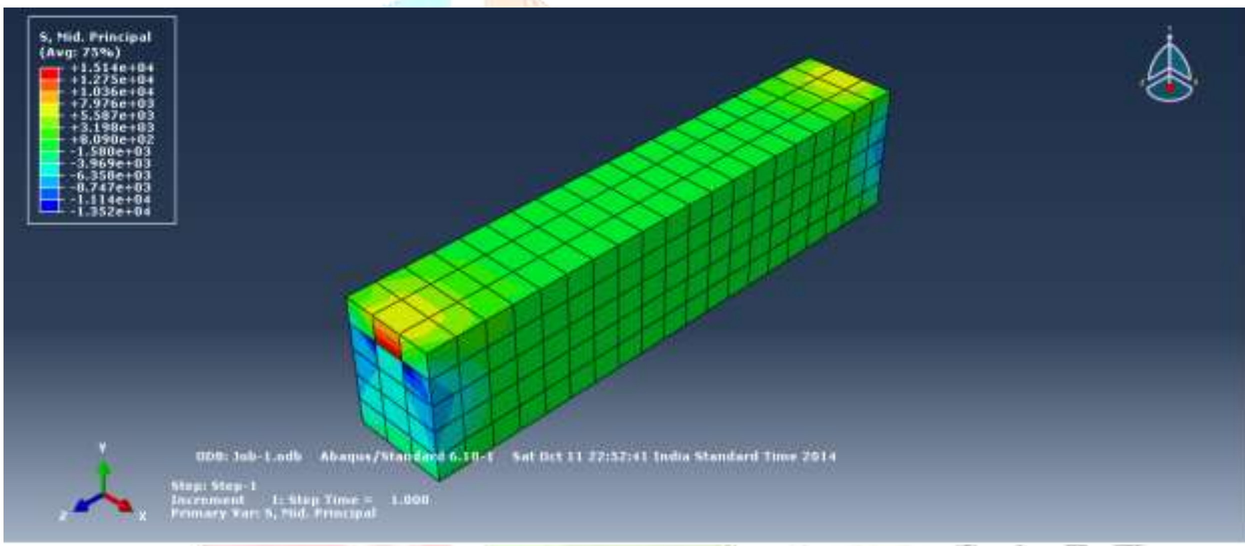


Figure 8. Average principal stress

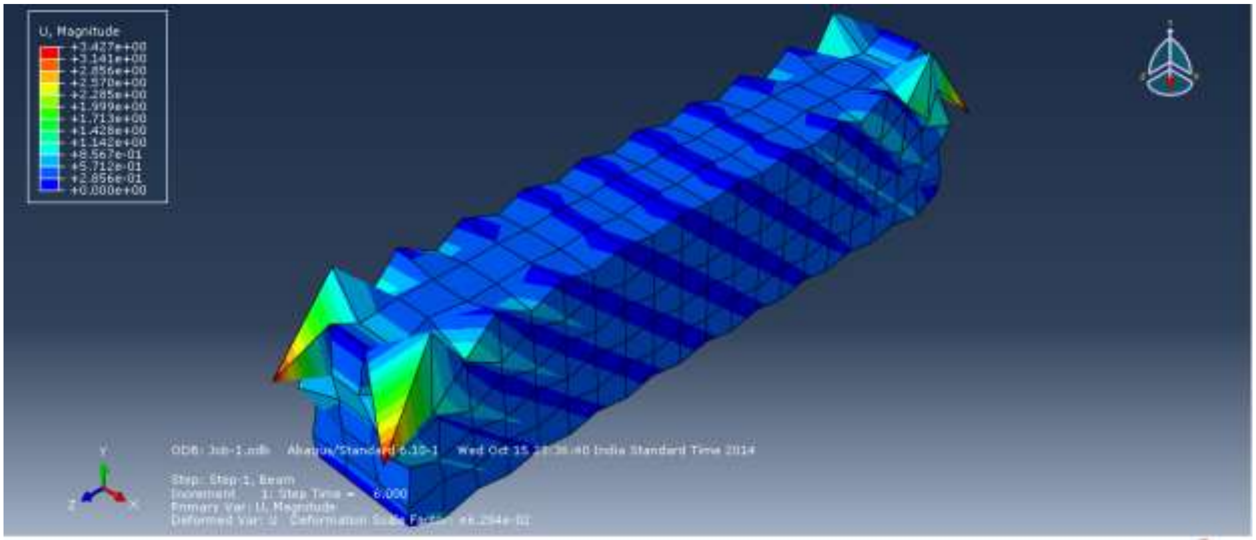


Figure 9. Deformation

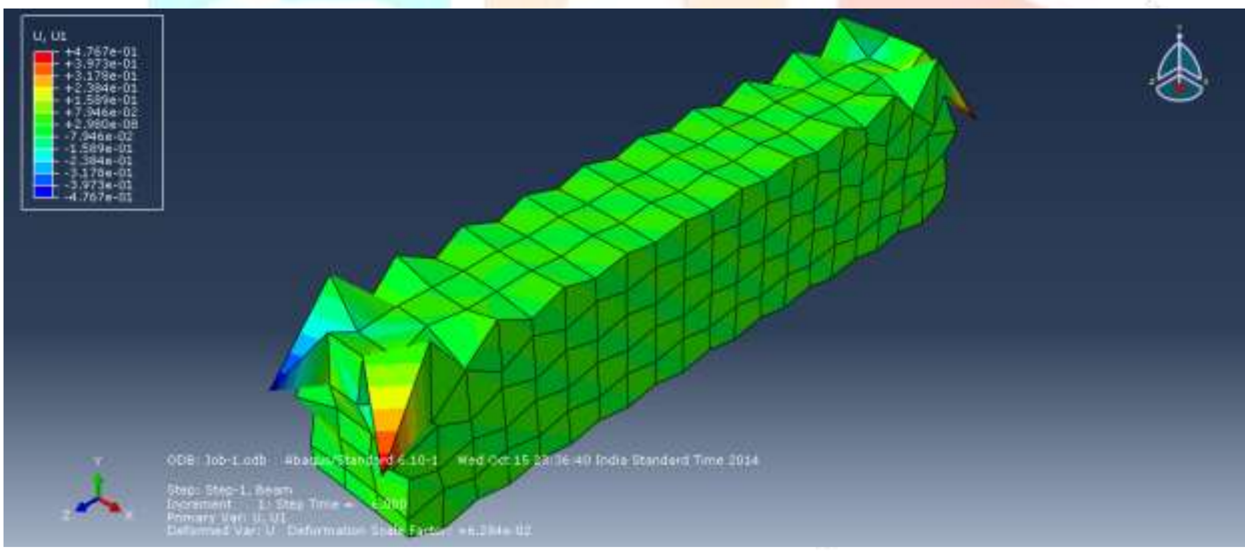


Figure 10. Deformation 1

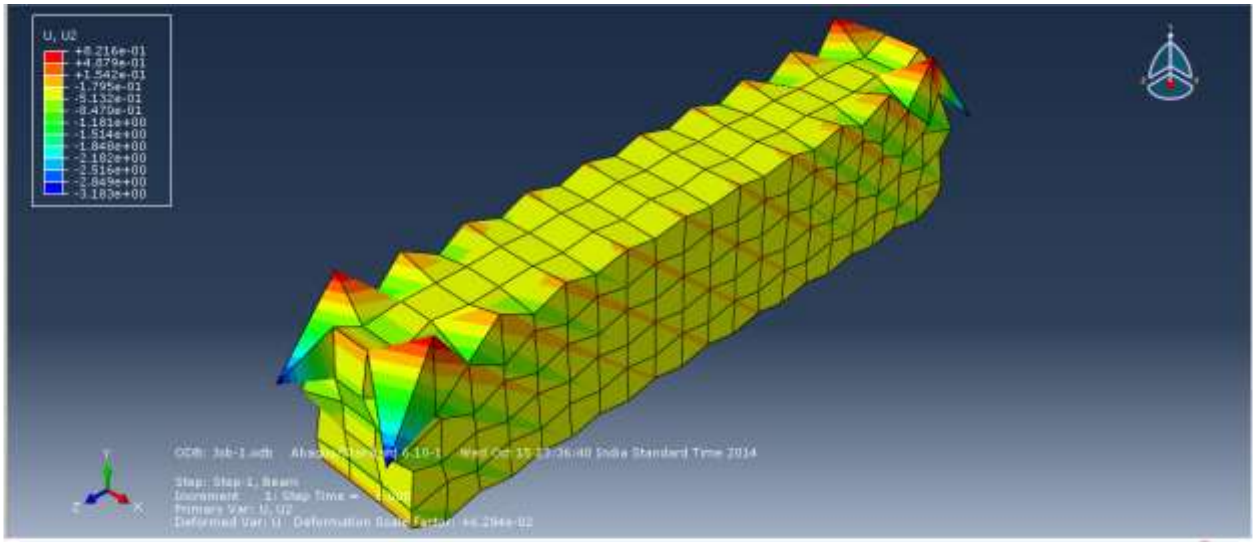


Figure 11. Deformation 2

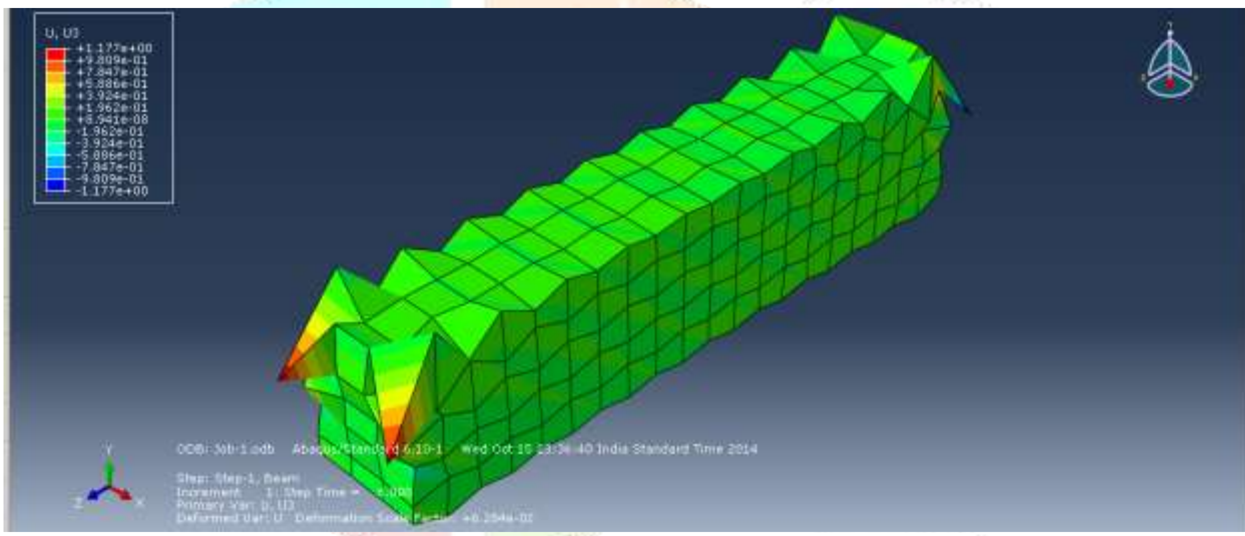
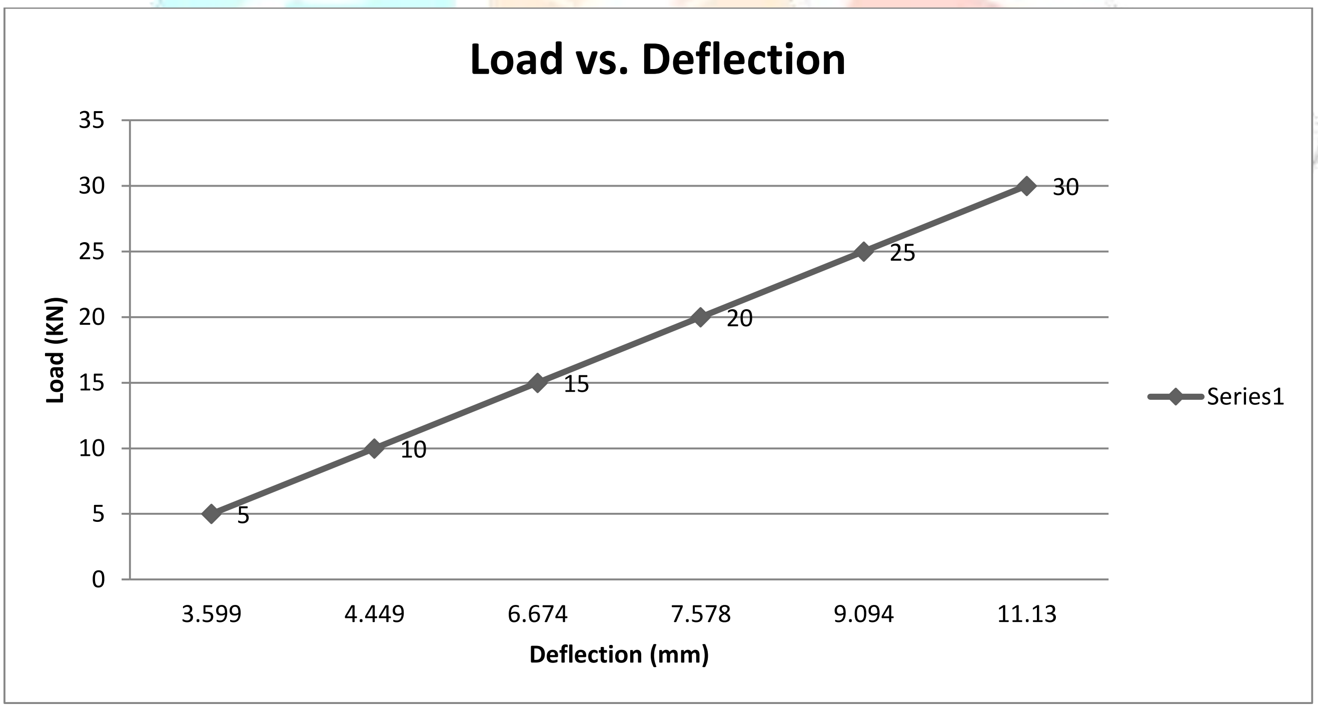


Figure 12. Deformation 3

UNIFORMLY DISTRIUTED LOAD AND CORRESPONDING RESULTS

S.No	Load in kN	Deflection in mm
1	5	3.599
2	10	4.449
3	15	6.674
4	20	7.578
5	25	9.094
6	30	11.130



Graph 2. Load vs. Deflection

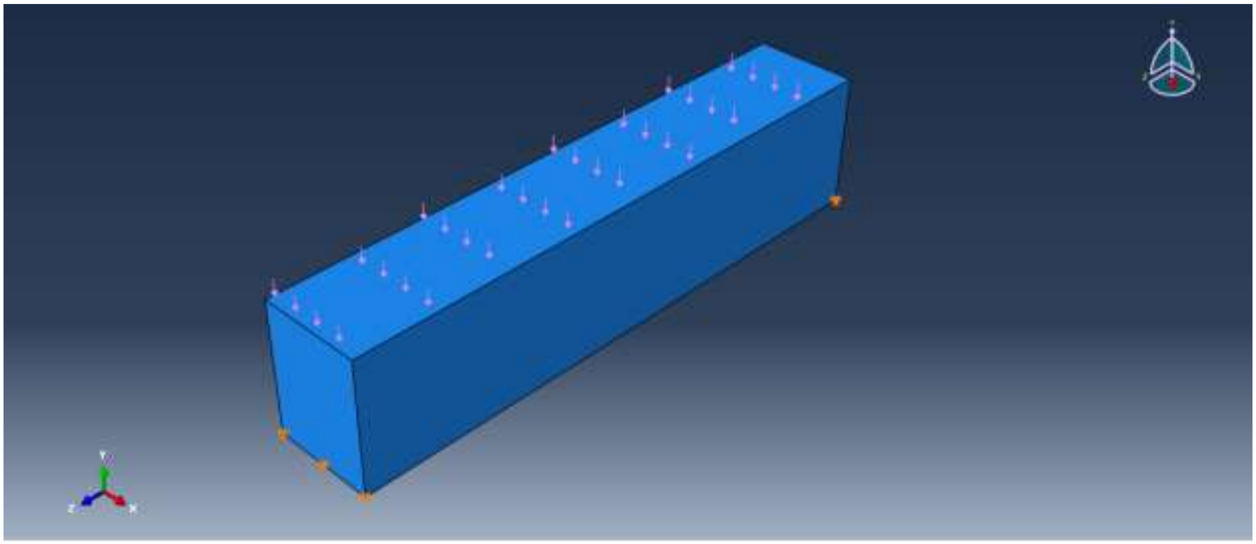


Figure 13. Simply supported beam with UDL

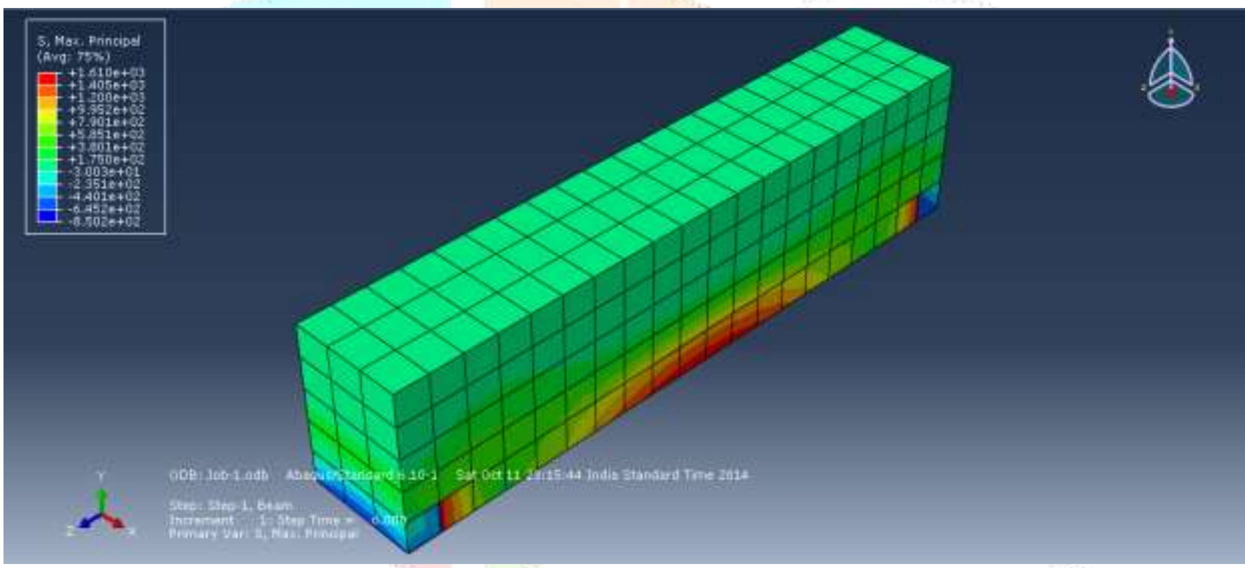


Figure 14. Maximum principal stress

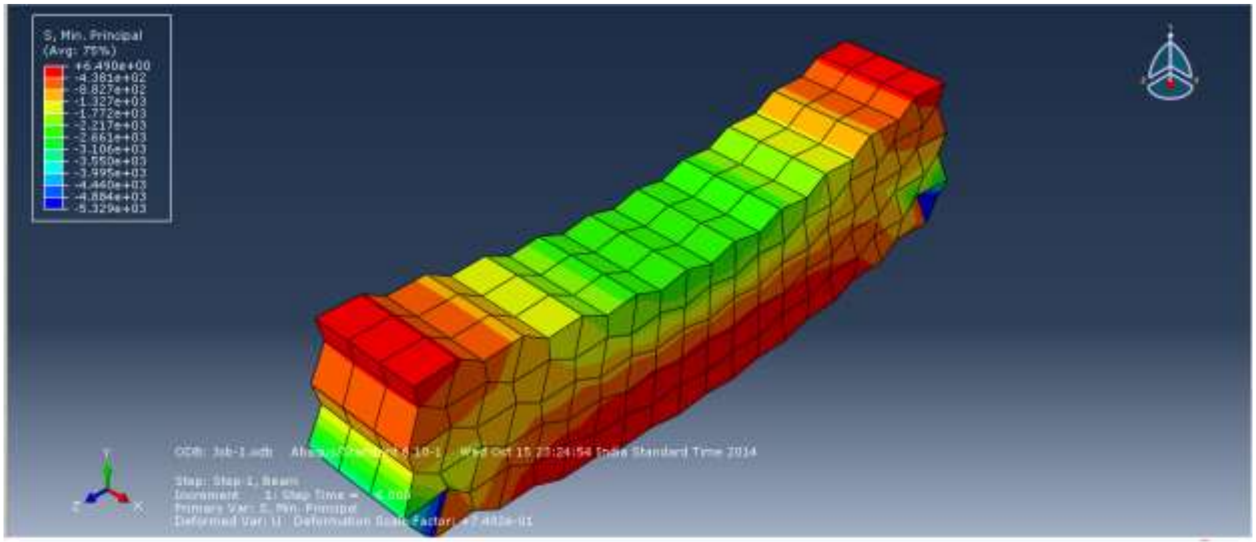


Figure 15. Minimum principal stress

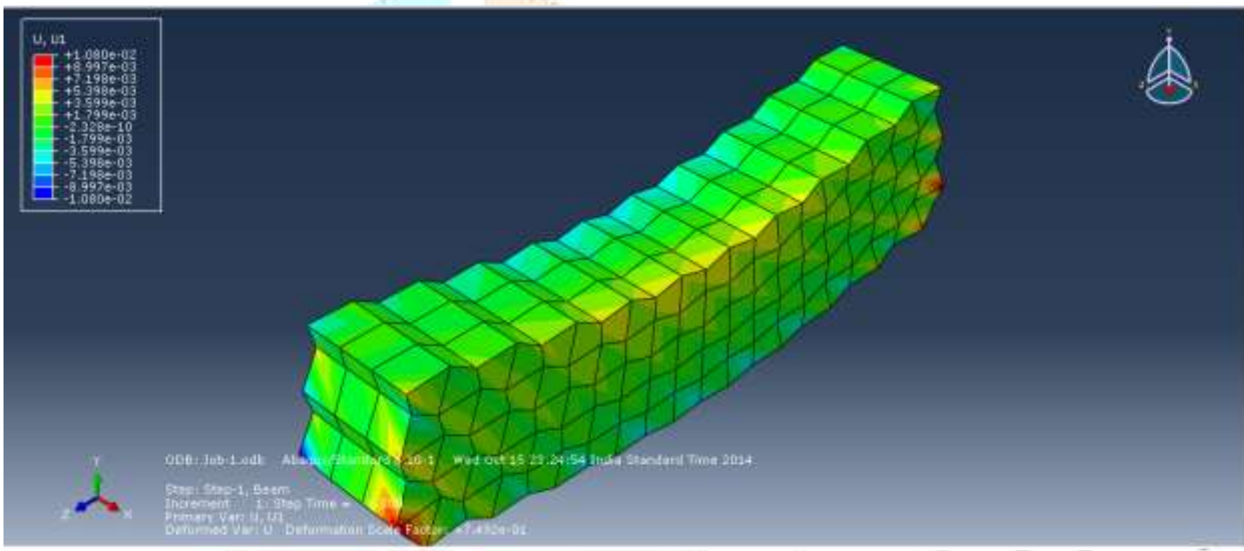


Figure 16. Deformation 1

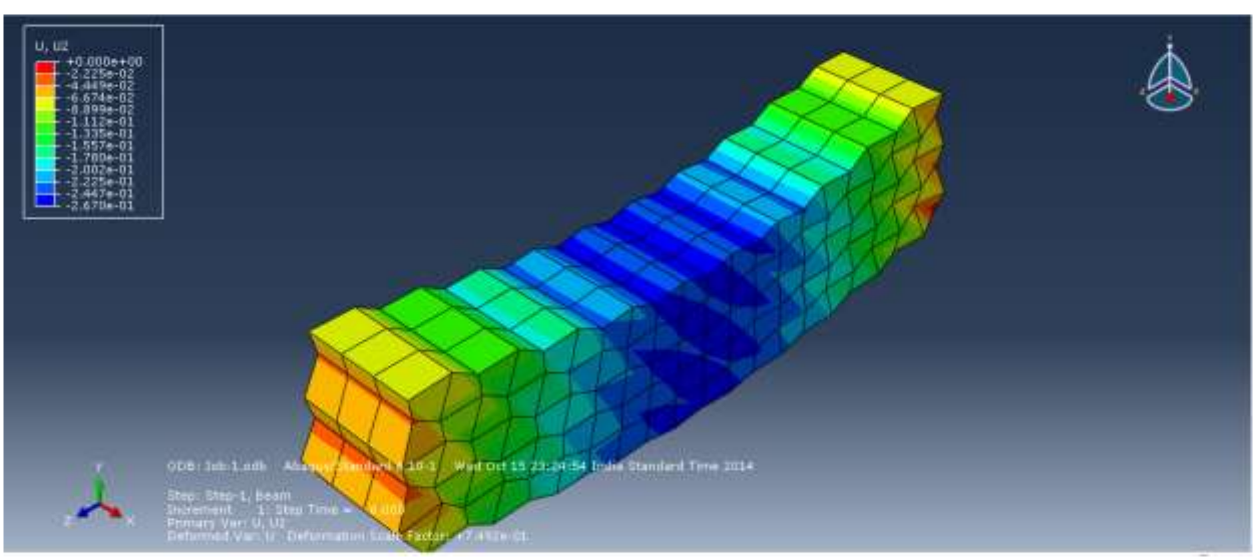


Figure 17. Deformation 2

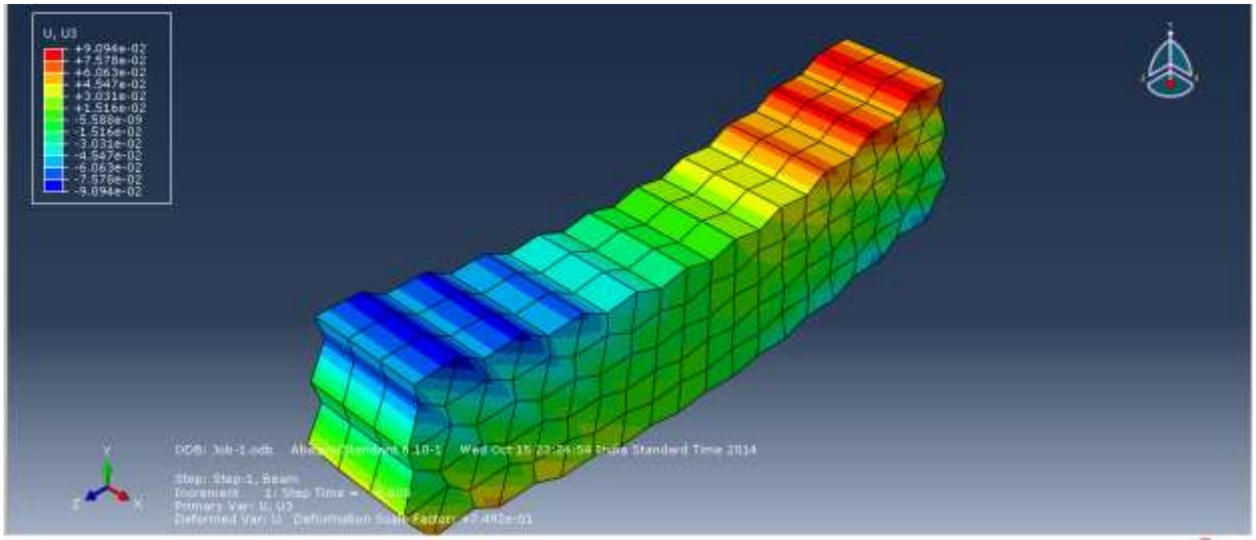


Figure 18. Deformation 3

VI. CONCLUDING REMARKS

• ABAQUS has many modeling characteristics with which to model reinforced concrete.

• ABAQUS can to do the following:

- model concrete and steel with beam and shell element
- simulate their interaction
- apply loads
- Calculate accurate results and predict behavior not generally obtained through experimentation.

• The accuracy of the model was validated, and the limitations of matching finite element models to experimental tests held under conditions that are less than ideal was illustrated.

• The development of a finite element model of an entire bridge illustrates not only the capability of ABAQUS to represent the behavior of a realistic structure but also the specific capability of the model to predict deflections, strains, and stresses while minimizing unnecessary complexities.

VII. REFERENCES

1. ABAQUS, (2004), Example problems manual, Version 6.4.1, HibbittKarlsson& Sorensen, Inc., USA.
2. ABAQUS/CAE, (2004), User's Manuals, Version 6.4.1, HibbittKarlsson& Sorensen, Inc., USA.
3. ABAQUS/Explicit, (2004), Theory Manual, Version 6.4.1, HibbittKarlsson& Sorensen, Inc., USA.
4. ACI 318-02/318R-02, (2002), Building Code Requirements for Structural Concrete and Commentary, American Concrete Institute, Farmington Hills, MI.
5. ASCE 7-02, (2002), Minimum Design Loads for Buildings and Other Structures, American Society of Civil Engineers, Reston, VA.
6. Hakuno, M., and Meguro, K., (1993), Simulation of Concrete-Frame Collapse Due to Dynamic Loading, Journal of engineering mechanics, 119(9), pp1709-1723.
7. Kishi, N., (2004), Practical Methods for Impact Test and Analysis, Structural Engineering Series, JSCE, impact problems, No.15 (in Japanese).
8. Leyendecker, E. V., and Burnett, E. F. P., (1976), the Incidence of Abnormal Loading in residential Buildings, National Bureau of Standards, NBS 98, and Washington, D.C.
9. Leyendecker, E. V., and Ellingwood, B. R., (1977), Design Methods for Reducing theRiskof Progressive Collapse in Buildings, National Bureau of Standards, NBS 98, and Washington, D.C.
10. Luccioni, B. M., Ambrosini, R. D., andDanesi, R. F., (2004), Analysis of building Collapse under blast loads, Journal of engineering structures, ASCE, 26, pp 63-71.

11. A.R.Santhakumar, concrete technology, oxford university press, 2007.

12. Alfaiate, J., Pires, E.B., & Martins, J.A.C. (1997). A Finite Element Analysis of Non-Prescribed Crack Propagation in Concrete. Computers & Structures, 63(1): 17-26.

13. Ashour, A.F., & Morley, C.T. (1993). Three-dimensional Nonlinear Finite Element Modeling of Reinforced Concrete Structures. Finite Elements in Analysis and Design, 15:4355.14. Azizinamini, A. (1994). Old Concrete Slab Bridges: Analysis. ASCE Journal of Structural Engineering, 120(1): 3305-3319.

15. Barzegar, F. (1994). Generating Reinforcement in FE Modeling of Concrete Structures ASCEJournal of Structural Engineering, 120(5): 1656-62.