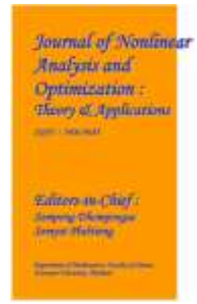
Journal of **N**onlinear **A**nalysis and **O**ptimization
Vol. 15, Issue. 1 : 2024
ISSN : **1906-9685**



Analysis of Simply Supported Beam using ABAQUS Software

Sudhir Vummadisetti[1*], Durga Kunal Kolla[2], Aakash Karise[3], Anand Prasad[4], Ramesh Gorli[5], Meenakshi Goddu[6]

Associate professor[1], UG student[2,3,4,5]
sudhir.civilviit@gmail.com[1*] (corresponding author)

Vignan's Institute of Information Technology (A),
Duvvada, Visakhapatnam, Andhra Pradesh

ABSTRACT

There are numerous requests for ecologically neighborly materials these days, which would incredibly diminish the reliance on engineered filaments and their composites. Jute fiber might perform well in put of engineered fiber to fortify composites. The concept of utilizing fiber fortified polymer could be a characteristic sort of composite that coordinating a strong quality for different applications and operations of building ventures, or for the improvement prepare of auxiliary applications. The essential basis for utilizing fiber fortified polymer composites is that they are natural, effortlessly open, renewable, and naturally neighborly. They moreover decrease the abuse of non- renewable assets, which makes a difference to avoid natural contamination and the vitality utilization of non-biodegradable development fabric components. The jute fiber fortified polymer (JFRP) is being utilized in this investigate to discover their mechanical conduct. Utilizing examples made from jute fiber, the ductile and flexural characteristics are evaluated. Steel bars and wires that are used to strengthen and support concrete in buildings are called steel reinforcement. Steel bars and wires are put inside concrete structures to add to the load bearing capacity of the structure.

The nonlinear investigation of a fortified concrete beam was conducted based on the limited component examination computer program ABAQUS. In this essentially bolstered pillar investigation, the versatility show of concrete harm in ABAQUS has been presented altogether. At long last, the comes about of the experimentation and the ABAQUS examination were compared in a graph, appropriately reasons of the result contrast between the two strategies were talked about, which can be a valuable reference for the assist think about of the nonlinear examination of fortified concrete. the Examination of the JFRP bar & steel bars reinforced beam

is conducted in ABAQUS Computer program & the comparison of them Uncovers that JFRP bar tends to have more strength, Quality & Versatile preferences than the steel fortification bar.
Keywords: Jute fiber reinforced

1. INTRODUCTION

FRP called a fiber reinforced concrete. While firstly defining about composites, a composite has different type of materials that do not blend and dissolve into each other, and to describe in brief (FRP) is a composite material consists of fibers of significant strength and stiffness embedded in matrix with distinct boundaries between them, as described before material consists of 2 components, fibres are usually like natural fibres of (jute, hemp, coir), glass and carbon or aramid. And matrix is usually an epoxy, vinyl ester or polyester etc. Both fibres and matrix maintain their physical and chemical identities but their combination performs a function that cannot be done by each constituent separately

Steel bars and wires that are used to strengthen and support concrete in buildings are called steel reinforcement. Steel bars and wires are put inside concrete structures to add to the load bearing capacity of the structure. The nonlinear analysis of a reinforced concrete beam was conducted based on the finite element analysis software ABAQUS. In this simply supported beam analysis, the plasticity model of concrete damage in ABAQUS has been introduced thoroughly. Finally, the results of the experimentation and the ABAQUS analysis were compared in a diagram, accordingly reasons of the result difference between the two methods were discussed, which can be a useful reference for the further study of the nonlinear analysis of reinforced concrete.

The comparison on the elastic properties of jute FRP beam and the steel reinforcement beam is carried out in the ABAQUS software. The strength characteristics of jute FRP is more admirable compared to steel reinforcement and more durable with corrosion resistance. The high cost of the jute FRP is the demerit compared to the steel reinforcement.

2. EXPERIMENTAL INVESTIGATION

M20 concrete mix design was completed in accordance with IS 10262-2001 and IS 10266-20019. For a period of 28 days, six cubes were tested and cast. For casting, concrete of grade M20 was utilized. Three cubes were tested to determine the concrete's initial strength after seven days of curing, and the results showed that it was 65% of the design value. After 28 days of curing, the remaining 3 cubes underwent the same process. Following a 28-day curing period, the beams were experimentally loaded twice using a hydraulic actuator, as shown in Figure 2A. To achieve an effective span of 3000 mm, a roller on one side and a hinge on the other were employed in a simply supported configuration. The 230 x 300 mm test specimen was fitted with the hydraulic actuator in its center after being set up on the supports. Between the load jack and the test specimen, an intermediate beam with two roller supports on the top surface of the specimen was positioned for two-point load action.

2.1 Beam casting and testing Reinforced concrete beam was designed as per limit state method using IS 456: 2000 for mild exposure condition. The reinforcement provided in the beam as given below and detailing is shown in Figure 1.

Tension reinforcement: 2 nos. of 12 mm diameter Fe 415 steel.
Transverse reinforcement: 2 legged 8 mm diameter stirrups at 150 mm c/c
Cover to concrete: 40 mm

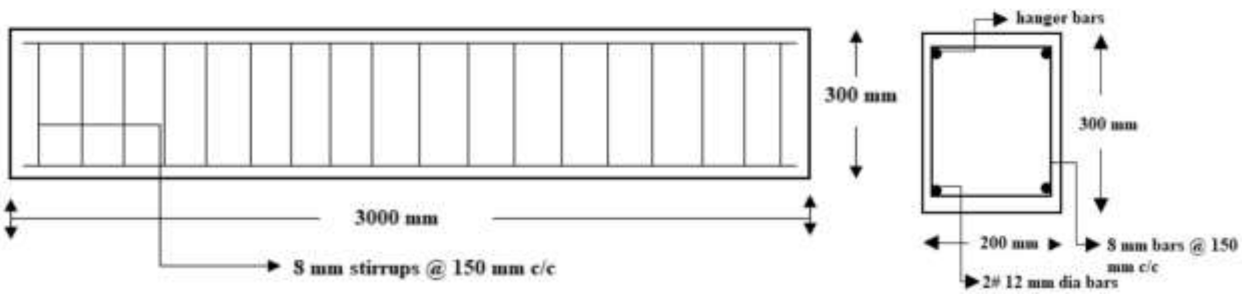


FIGURE 1: DETAILING OF RC BEAM SPECIMEN

Jute NSM of 20mm thick is provided in another beam with the dimensions and reinforcement as of RC beam.

Following a 28-day curing period for the beam, the specimen underwent two-point hydraulic loading Jacks. For an effective span of 3000 mm, a simply supported condition was employed, with a hinge on one side and a roller on the other. The test specimen, which measured 230 by 300 mm, was positioned on the supports, and the hydraulic jacks were positioned in the test specimen's center.

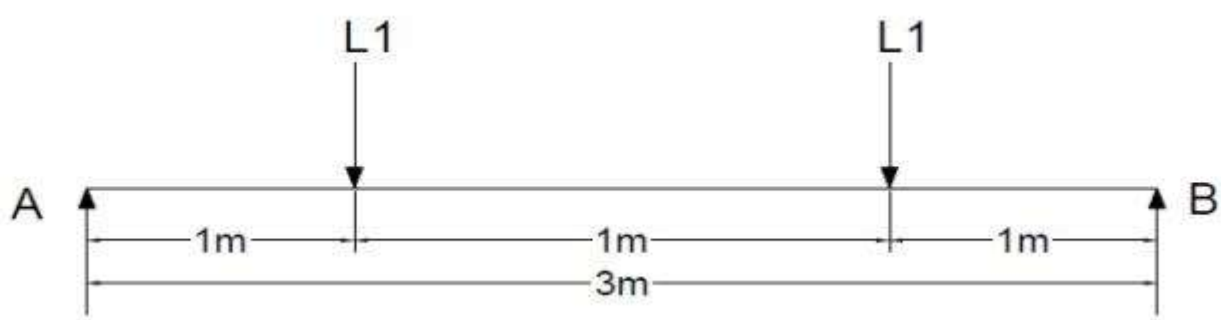


FIGURE 2: TWO POINT LOADING

2.2 DISPLACEMENT

The deflection of beam may be generally limited to the following:

The final deflection due to all loads including the effects of temperature, creep and shrinkage and measured from the base level of the supports of Floors, roofs and all other horizontal members, should not normally exceed span / 250.
Here,
Span = 3000mm
Permissible deflection = 3000/250 = 12 mm

3. ABAQUS MODELLING

A software suite for finite element analysis, ABAQUS was first released in 1978. This software is available for purchase and can be used for both linear and nonlinear analysis. A three-dimensional finite element model of a reinforced concrete beam was created based on the primary goals of this study.

3.1 Creating part

The beam is modeled as a two-dimensional profile of a deformable solid body.
Next, extrusion was used to transform it into three dimensions. The models BEAM_CONCRETE, REBAR_STEEL, and STIRUPPS_STEEL have three additional parts generated in them. In Figure 5, the extruded portion is displayed. In this work, steel is modeled as a two-noded beam element and concrete as an eight-noded element.

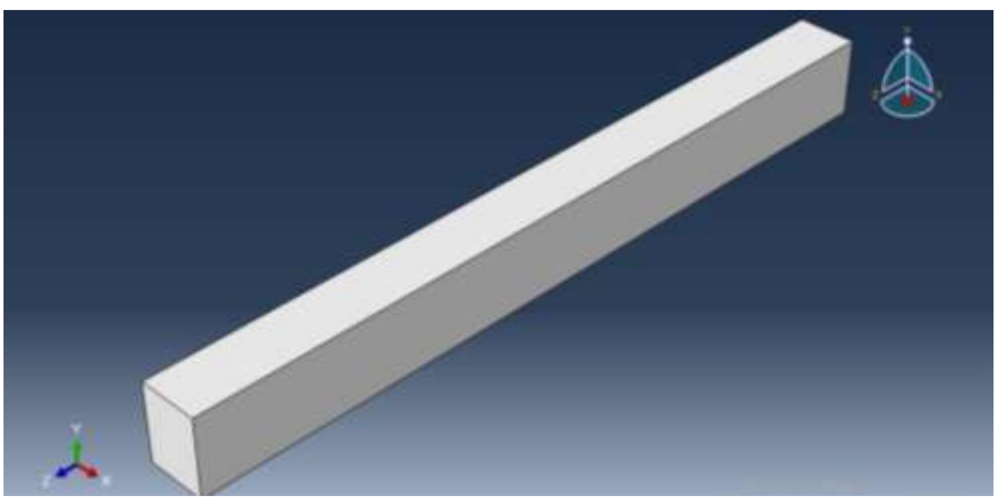


FIGURE 3: EXTRUDED PART

A 3 m long, 230 mm wide, and 300 mm deep beam is formed. Since a beam's length is comparatively greater than its breadth, plane stress elements are frequently employed to simulate beams in two dimensions. Both 1D and 2D elements can be used to model the reinforcement as embedded reinforcement. The rebars were represented in the model as two-node beam elements joined to the nodes of nearby solid components.

3.2 Material property

The property of material.

1.Young’s modulus of 22360 MPa for concrete and Young’s modulus of 215000 MPa for steel.

2. Poisson’s ratio of 0.3 for steel and 0.2 for concrete

3. Mass Density of 2400 kg/m3 for concrete and Mass Density of 7460 kg/m3 for steel.

3.3 Assigning section

The beam is given a homogeneous solid segment that has been formed. The two nodded beam elements made for reinforcements were assigned to steel, while the solid section constructed in the previous phase was assigned to concrete.

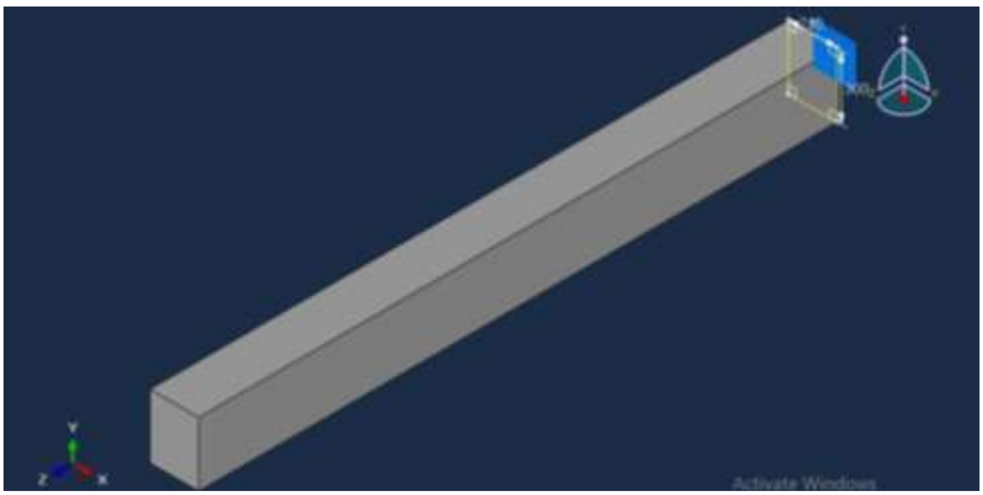


FIGURE 4: MATERIAL PROPERTY ASSIGNED SECTION

In this simulation, a pressure load is applied over the beam top, causing the simply supported beam to respond statically. As a result, there will be two steps in this model, and the first is assembly.

• The first stage involves applying the boundary conditions that limit the simply supported beam and two.

• A generic, static analysis step is used, applying pressure load to the beam's upper face.

3.4 Step creation and assembling the model

In this simulation, a pressure load is applied over the beam top, causing the simply supported beam to respond statically. As a result, there will be two steps in this model, and the model is built.

- The first stage involves applying boundary conditions to confine both the simply supported beam and two more beams.
- Applying a generic static analysis step with a pressure load on the beam's upper face.

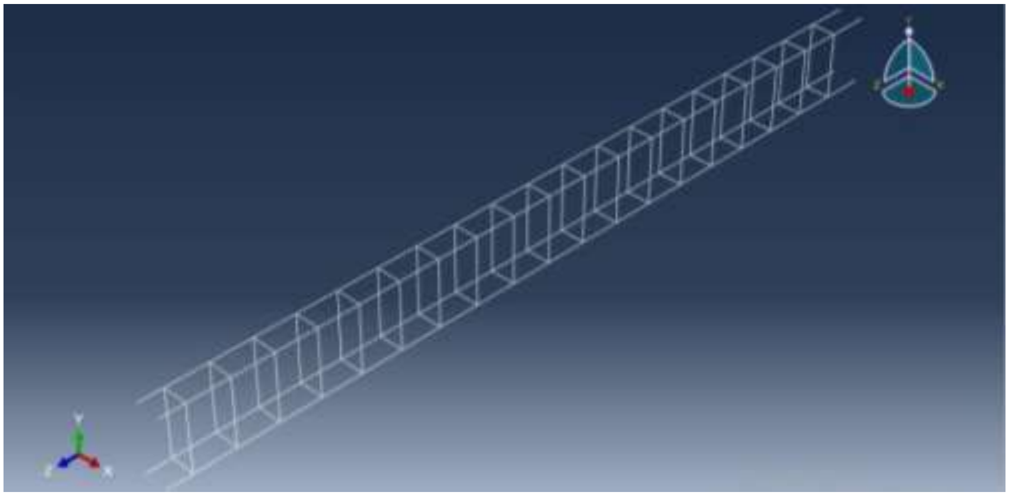


FIGURE 5: ASSEMBLING THE MODEL

- In the jute NSM beam model we placed jute FRP of 20mm at the bottom

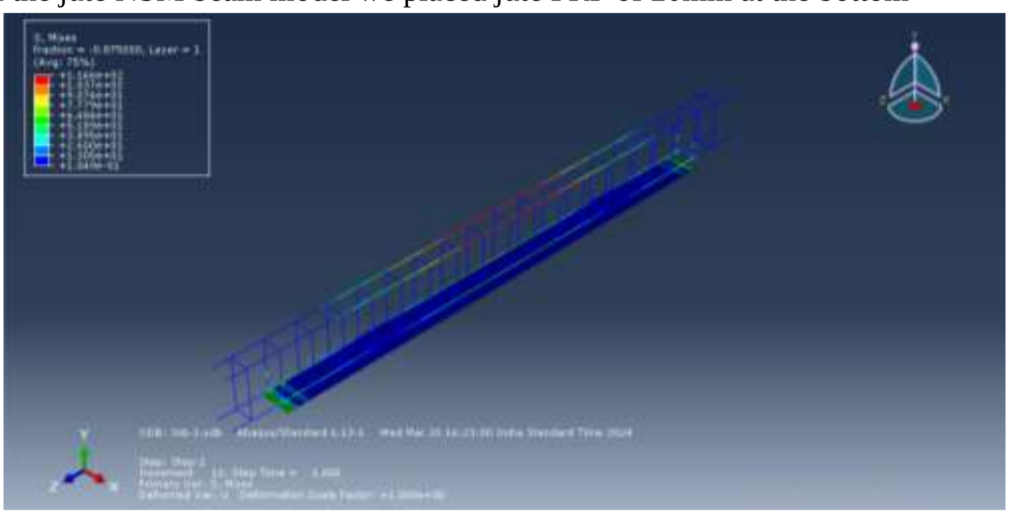


FIGURE 6: JUTE NSM ASSEMBLING

3.5 Applying boundary condition and loading on the model

The load dialog box opens and a new load is made in the model tree. The load option in the dialog box is POINT LOAD. In step 1, the load will be chosen and applied. Pressure is delivered to the beam in the chosen Step list, where the point load is dispersed across the surface area of the beam in contact. The load magnitude of -50000 (CF2) has been chosen, and the load is applied along the Y-axis as indicated by the negative sign.

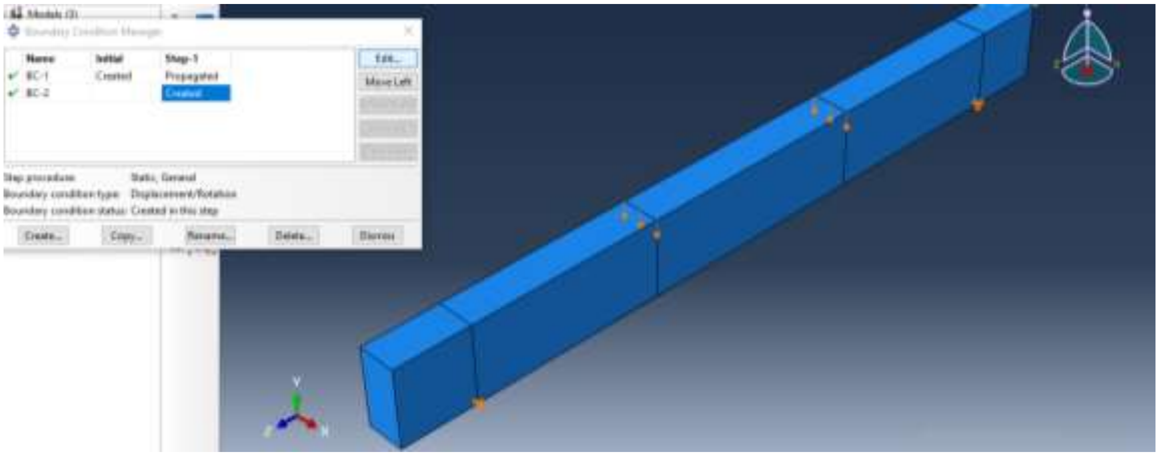


FIGURE 7: APPLYING BOUNDARY CONDITIONS

3.6. Meshing the model

Meshing is the process of creating nodes and elements. The process of creating nodes and joining them to create a mesh defines elements. The mesh module is created via finite element meshing. The element form, element type, and meshing mechanism can all be chosen within ABAQUS. In ABAQUS, a variety of meshing strategies are available to find the most convergent solution. The color of the model indicates which meshing technique is used by default. When the model shows an orange color, it means that the user must help it to be meshed.

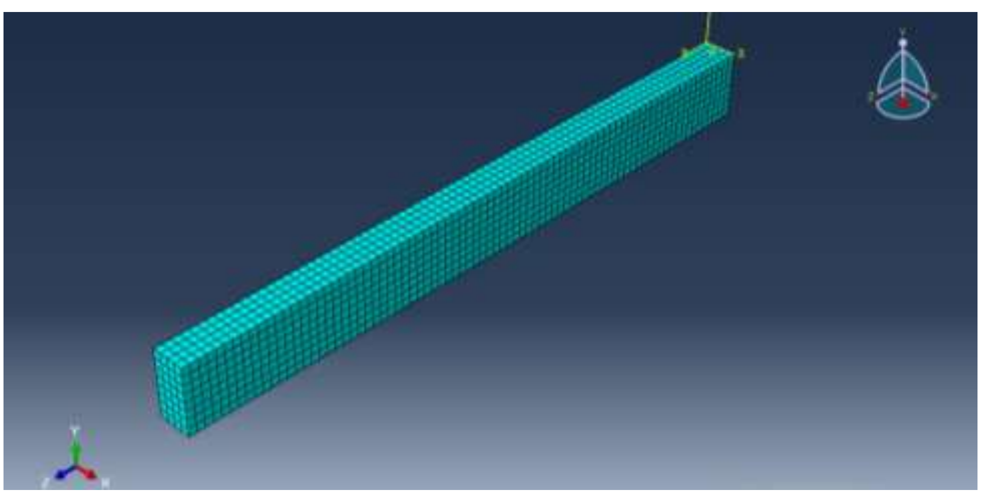


FIGURE 8: MESHED MODEL

3.7. Creating and submitting an analysis

Every property has been specified, assigned, and meshing when the model is constructed. After that, the analysis is performed on the model.

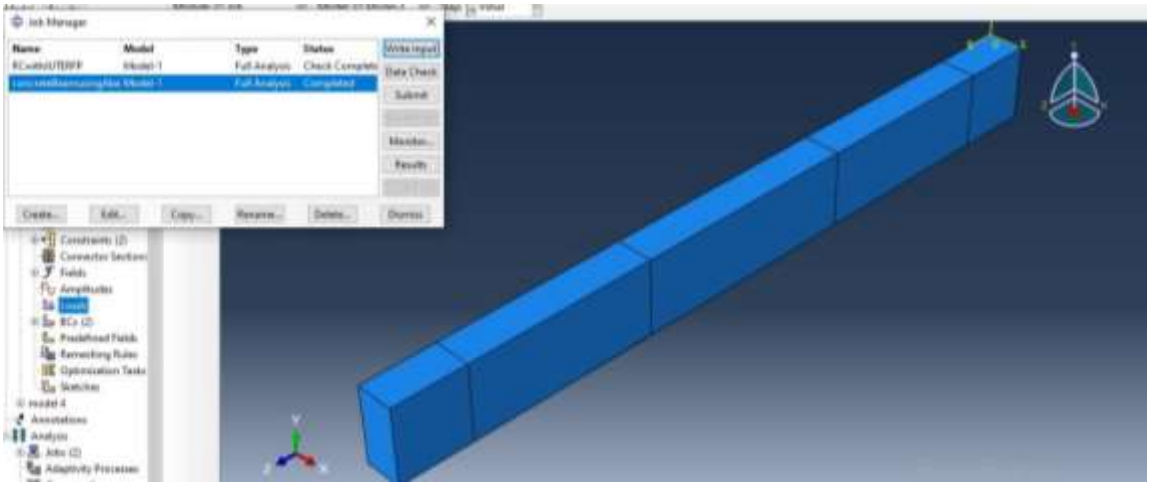


Figure 9: Run analysis

4. EVALUATION OF RESULTS

By giving the model the appropriate qualities, the beam was examined. The error-free model is indicated by the analysis that does not terminate. The Visualization module displays the analysis's findings. The outcomes of the experiment and analysis were then compared with the deflection results.

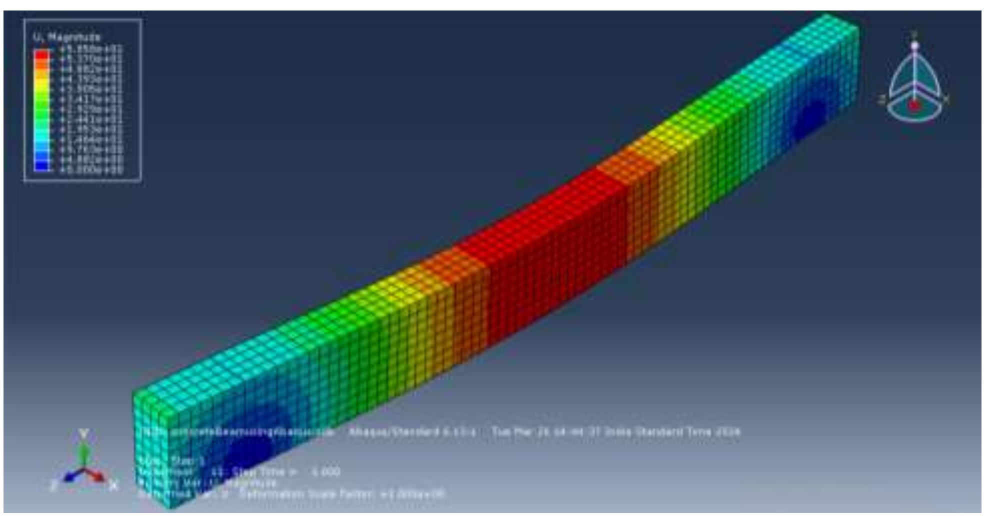


FIGURE 10: DEFLECTED BEAM

5. COMPARISON OF RESULTS

The theoretical and experimental values as stated in the paper were compared with the ABAQUS-analyzed beam. The parameters assigned for both concrete and steel in the model behaved well, and the results were determined to be in line with the theoretical calculation. The results are summarized in Table 2. After that, the model was given two new assignments for stress reinforcement: two numbers of 16 mm diameter bars and two numbers of 20 mm diameter bars. The beam has produced the anticipated outcomes in both situations.

BEAM ID	BEAM 1	BEAM 2	BEAM 3
Beam Size (mm)	230 x 300 x 3000	230x300x3000	230x300x3000
Diameter of rod	12	16	20
A_{st} of rod mm^2	226.1	402.1	628.3
Theoretical displacement (mm)	4.14	4.14	4.14
Numerical displacement (MPa)	3.925	3.743	3.557
Experimental strength (mm)	2.76	2.81	2.91
Permissible displacement (mm)	12	12	12

TABLE 1: RC BEAM NUMERICAL ANALYSIS VALUES

BEAM ID	Jute NSM 1	Jute NSM 1	Jute NSM 1
Beam Size (mm)	230 x 300x 3000	230x300x3000	230x300x3000
Diameter of rod	12	16	20
A_{st} of rod mm^2	226.1	402.1	628.3
NSM Thickness (mm)	20	20	20
Theoretical displacement (mm)	4.06	4.06	4.06
Numerical	3.253	3.104	3.012

BEAM ID	Jute NSM 1	Jute NSM 1	Jute NSM 1
Beam Size (mm)	230 x 300x 3000	230x300x3000	230x300x3000
Diameter of rod	12	16	20
displacement (MPa)			
Experimental strength (mm)	2.98	3.01	3.10
Permissible displacement (mm)	11	11	11

TABLE 2: JUTE NSM BEAM NUMERICAL ANALYSIS VALUES

6. CONCLUSIONS

The below conclusions were derived from the above study:

1. Theoretical displacement is less compared to the nominal beams. The average Theoretical displacement is reduced by 1.93%.
2. Numerical displacement is less compared to the nominal beams. The average Numerical displacement is reduced by 16.50%.
3. The experimental strength is high compared to the nominal beams. The average experimental strength is increased by 7.20%.
4. The Permissible displacement is less compared to nominal beams. The average permissible displacement is reduced by 8.33%.

REFERENCES:

1. T.S.Vishnu Kumar, N.Rushwanth Chowdary, "FINITE ELEMENT MODELING OF RCC BEAM BY USING ABAQUS", International Journal of Creative Research Thoughts (IJCRT), Volume 8, Issue 7 July 2020.
2. Dimosthenis Floros Olafur Agust Ingason, "Modelling and simulation of reinforced concrete beams" Department of Applied Mechanics Division of Solid Mechanics Chalmers University Of Technology, 2013.
3. T. Tejaswini, Dr.M.V.Rama Raju, "Analysis of RCC Beams using ABAQUS", International Journal of Innovations in Engineering and Technology (IJIET), Volume 5 Issue 3, June 2015.
4. A.Hemamathi, Sanjith Murugavel , Binu Sukumar , M.Usha Rani, "Numerical Investigation of Precast Grouted Sleeve Connection under Cyclic Loading Using ABAQUS", Journal of Physics: Conference Series, 2070 (2021) 012178.
5. Sayed M S and Sajjad S R, "Analytical Study of Reinforced Concrete Beams Tested under Quasi-Static and Impact Loadings", Applied Science, 2019.

6. S.V.Chaudhari, M.A.Chakrabarti, “Modeling of concrete for nonlinear analysis Using Finite Element Code ABAQUS”, International Journal of Computer Applications (0975 – 8887), Volume 44– No.7, April 2012.
7. Sayed Mohamad Soleimani and Sajjad Sayyar Roudsari, “Analytical Study of Reinforced Concrete Beams Tested under Quasi-Static and Impact Loadings” Journal of Applied Sciences, Volume 9, 2019.
8. Hamid S, Mahdi S, Amir H A, Mohammad A and Ali S, “Evaluation of reinforced concrete beam behaviour using finite element analysis by ABAQUS”, Scientific Research and Essays Vol. 7(21), pp. 2002-2009, 7 June 2012.
9. I. Saifullah, M. Nasir-uz-zaman, S.M.K. Uddin, M.A. Hossain and M.H. Rashid, “Experimental and Analytical Investigation of Flexural Behavior of Reinforced Concrete Beam”, International Journal of Engineering & Technology IJET-IJENS Vol: 11 No: 01, February 2011.
10. Swoo H L, Ali A, Kyung J, Hee D, “ABAQUS modeling for post-tensioned reinforced concrete beams”, Journal of Building Engineering Volume 30, July 2020.
11. Workshop 1, “Linear Static Analysis of a Cantilever Beam, Introduction to Abaqus”, D’s Simulia
12. Indian Standard Plain and Reinforced Concrete - Code of Practice, Is. 456: 2000.
13. Indian Standard Concrete Mix Proportioning – Guidelines (Second revision) IS 10262: 2019.
14. Indian Standard Methods of Tests for Strength of Concrete IS 516: 2004. [15] Dassault Systems “Getting Started with Abaqus/CAE Abaqus 2016”.
15. Keshav, Vasanth, and Sudhir Vummadisetti. "Non-rectangular plates with irregular initial imperfection subjected to nonlinear static and dynamic loads." International Journal of Advances in Engineering Sciences and Applied Mathematics 15, no. 4 (2023): 155-158.
16. Vummadisetti, Sudhir, and S. B. Singh. "The Influence of Cutout Location on the Postbuckling Response of Functionally Graded Hybrid Composite Plates." In Stability and Failure of High-Performance Composite Structures, pp. 503-516. Singapore: Springer Nature Singapore, 2022.
17. Sathi, Kranthi Vijaya, Sudhir Vummadisetti, and Srinivas Karri. "Effect of high temperatures on the behaviour of RCC columns in compression." Materials Today: Proceedings 60 (2022): 481-487.
18. Vummadisetti, Sudhir, and S. B. Singh. "Buckling and postbuckling response of hybrid composite plates under uniaxial compressive loading." Journal of Building Engineering 27 (2020): 101002.
19. Vummadisetti, Sudhir, and S. B. Singh. "Postbuckling response of functionally graded hybrid plates with cutouts under in-plane shear load." Journal of Building Engineering 33 (2021): 101530.
20. Vummadisetti, S., and S. B. Singh. "Boundary condition effects on postbuckling response of functionally graded hybrid composite plates." J. Struct. Eng. SERC 47, no. 4 (2020): 1-17.

21. Singh, Shamsher Bahadur, Sudhir Vummadisetti, and Himanshu Chawla. "Development and characterisation of novel functionally graded hybrid of carbon-glass fibres." International Journal of Materials Engineering Innovation 11, no. 3 (2020): 212-243.
22. Singh, S. B., Sudhir Vummadisetti, and Himanshu Chawla. "Assessment of interlaminar shear in fiber reinforced composite materials." Journal of Structural Engineering 46, no. 2 (2019): 146-153.
23. Singh, S. B., Himanshu Chawla, and Sudhir Vummadisetti. "Experimental and Analytical Studies of Failure Characteristics of FRP Connections." In Recent Advances in Structural Engineering, Volume 2: Select Proceedings of SEC 2016, pp. 755-757. Springer Singapore, 2019.
24. Singh, S. B., Sudhir Vummadisetti, and Himanshu Chawla. "Influence of curing on the mechanical performance of FRP laminates." Journal of Building Engineering 16 (2018): 1-19.